403 Poyntz Avenue, Suite B
Manhattan, KS 66502
USA
+1.785.770.8511
www.thunderheadeng.com

User Manual

Pathfinder 2015

Disclaimer

Thunderhead Engineering makes no warranty, expressed or implied, to users of Pathfinder, and accepts no responsibility for its use. Users of Pathfinder assume sole responsibility under Federal law for determining the appropriateness of its use in any particular application; for any conclusions drawn from the results of its use; and for any actions taken or not taken as a result of analyses performed using these tools.

Users are warned that Pathfinder is intended for use only by those competent in the field of egress modeling. Pathfinder is intended only to supplement the informed judgment of the qualified user. The software package is a computer model that may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions by the model could lead to erroneous conclusions. All results should be evaluated by an informed user.

Acknowledgements

This work was originally made possible by a Small Business Innovative Research (SBIR) grant (2005-2007) by the United States National Science Foundation.

We would like to thank Rolf Jensen and Associates for their assistance with testing and other suggestions that helped guide the development of the simulator.

We would also like to thank the users whose feedback helps us improve the software and incorporate more useful features. The Pathfinder support forum can be found at www.thunderheadeng.com/pathfinder.

Trademarks

All other product or company names that are mentioned in this publication are tradenames, trademarks, or registered trademarks of their respective owners.

Table of Contents

1. Introduction 1
Graphical User Interface 1
Model Representation 3
Simulation Modes 4
Limitations and Known Issues 4
Simulator Name 4
System Requirements 4
Contact Us 6
2. Pathfinder Basics 7
Navigation View 7
3D and 2D Views 8
Navigating the 3D view 8
Navigating the 2D view 9
Resetting the view 9
Filling the view 9
Drawing in the 3D and 2D views 10
View Options 10
Render Options 10
Occupant Display 11
Coloring Rooms 11
Room Opacity 11
Model Organization with Groups 11
Creating sub-groups 12
Changing groups 12
3. Creating Movement Space 13
Floors 13
Automatically creating floors 13
Manually creating floors 15
Changing the active floor 15
Showing all floors 16
Floor Properties 16
Rooms 16
Adding new rooms 16
Drawing Plane 18
Thin walls 18
Splitting rooms 19
Separating and merging rooms 19
Room properties 20
Preventing room-crossing 21
Obstructions/Holes 22
Arbitrarily-shaped obstructions (desks, tables, etc.) 22
Thick walls 23
Doors 24
Thin doors 24
Thick doors 25

Door properties 26
Stairs 27
Stairs between edges 27
Stairs extending from one edge 28
Stair properties 30
Ramps 31
Escalators 31
Moving Walkways 31
Elevators 32
Creating elevators 32
Elevator representation 34
Elevator properties 35
Nominal load 36
Connecting/Disconnecting floors 36
Call Sets 37
Exits 37
Importing Files 38
Importing images 38
Importing CAD files 39
Importing Revit Files 41
Importing PyroSim and FDS files 42
Working with Imported Data 42
Working with Images 42
Working with 3D CAD, PyroSim, and FDS files 43
Working with 2D DXFs 44
Filling in missing pieces 45
Flattening and Z Location 46
Visual Characteristics 47
Materials 47
Reorganizing and Making Fast Edits 49
4. Creating Occupants 50
Profiles 50
Advanced Speed Properties 53
Seeds 57
Customizing Occupants 57
Behaviors 58
Creating a new Behavior 58
Adding actions 59
Goto Waypoint action 60
Goto Rooms action 60
Goto Elevators action 61
Wait action 61
Adding Occupants 62
Individual placement 62
Group Placement 62
Placement in rooms 64
Redistributing Profiles and Behaviors 64
5. Views and Camera Tours 66

Views 66
Creating a View 66
Recalling a View 66
Editing a View 67
Camera Tours 67
Creating Tours from Scratch 67
Creating Tours from Existing Views 68
Preview a Camera Tour 69
Editing a Camera Tour 69
Changing Views/Tours without Re-running Simulation 69
6. Editing and Copying Objects 71
Transforming and Copying Objects 71
Moving 71
Rotating 72
Mirroring 74
Manipulating Objects with Handles 75
Selecting and deselecting a handle 75
Editing a handle 76
Room handles 76
Thin door handles 76
Thick door handles 77
Stair and Ramp handles 77
Occupant handles 77
Waypoint handles 78
7. Model Analysis 79
Measuring Distances 79
Checking Connectivity 79
Checking in-use Objects 81
Warnings and Errors 81
8. Simulating 83
Parameters 83
Time Parameters 83
Output Parameters 83
Behavior Parameters 85
Starting and Managing a Simulation 86
Simulating via command-line 87
Stopping and Resuming a Simulation 87
Stuck Occupants 88
9. Results 89
Summary Report 89
Door History 90
Door Flowrate and Specific Flow 90
Door Usage 91
Room History 91
Occupants Summary 92
Occupant History 92
3D Results 93
Navigating through a model 93

Displaying geometry input 94
Displaying occupants 94
Selecting Occupants 95
Viewing multi-floor problems 95
Animation playback 96
Refreshing Results 97
Viewing occupant paths 97
Controlling drawing detail/speed 97
Showing Saved Views 98
Showing Camera Tours 98
Creating Movies 98
Showing FDS Results 100
Occupant Contours/Heat Maps 103
10. References 1
11. Index 2

Figures

Figure 1-1: An example of the graphical user interface. Model of the Theater de Vest created by Van Hooft Adviesburo. 1
Figure 1-2: An example of the 3D Results view, showing occupants gathering at a refuge area before proceeding to elevators. Transparency has been used to help view the occupants. 2
Figure 1-3: An example time history graph 2
Figure 1-4: The triangulated navigation mesh used to represent one floor in the model. 3
Figure 1-5: Models used for system requirements comparison 5
Figure 2-1: 3D and 2D views 8
Figure 2-2: Render options 10
Figure 2-3: Predefined groups 12
Figure 3-1: Floor Creation panel 13
Figure 3-2: Auto floor creation and sorting 14
Figure 3-3: Adding a new floor 15
Figure 3-4: Floor property panel 16
Figure 3-5: Drawing a room with the polygon tool 17
Figure 3-6: Drawing a room with the rectangle tool 17
Figure 3-7: Drawing plane property 18
Figure 3-8: Adding a thin wall to a room 18
Figure 3-9: Dividing a room 19
Figure 3-10: Special case for drawing a thin wall 19
Figure 3-11: Merging rooms 20
Figure 3-12: Separating a room 20
Figure 3-13: Room properties panel 20
Figure 3-14: Edit Speed Modifier dialog 21
Figure 3-15: Make Doors Oneway dialog 22
Figure 3-16: Creating an obstruction 23
Figure 3-17: Subtracting walls 23
Figure 3-18: Door tool property panel 24
Figure 3-19: Adding a thin door to connect two touching rooms 25
Figure 3-20: Adding a thick door in the empty space between two rooms 26
Figure 3-21: Door property panel 26
Figure 3-22: Scripted Events dialog 27
Figure 3-23: Stair geometry requirements 27
Figure 3-24: Property panel for the two-point stair tool 28
Figure 3-25: Drawing stairs with the two-point stair tool 28
Figure 3-26: Property panel for the one-point stair tool 29
Figure 3-27: Drawing stairs using the one-point stair tool 29
Figure 3-28: Stair property panel 30
Figure 3-29: Stair door property dialog 30

Figure 3-30: Changing a stair into an escalator 31
Figure 3-31: Creating an Elevator 33
Figure 3-32: New Elevator dialog 34
Figure 3-33: Elevator representation 34
Figure 3-34: Elevator property panel 35
Figure 3-35: Elevator Priority dialog 36
Figure 3-36: Elevator Levels dialog 36
Figure 3-37: Elevator call sets 37
Figure 3-38: An exit door 37
Figure 3-39: Importing a background image 39
Figure 3-40: Dxf unit dialog box 40
Figure 3-41: Drawing rooms over a background image 42
Figure 3-42: Property panel for the floor extraction tool 43
Figure 3-43: A room extracted from a PyroSim file 43
Figure 3-44: Room extraction results from an imported 2D Floorplan 45
Figure 3-45: Using the door tool to fill a doorway 45
Figure 3-46: Set Z dialog 46
Figure 3-47: Imported Geometry property panel 47
Figure 3-48: Material dialog 48
Figure 4-1: The Edit Profiles dialog 50
Figure 4-2: An example of the 3D model dialog. 51
Figure 4-3: Editing advanced speed properties 54
Figure 4-4: Advanced Speed Properties dialog 55
Figure 4-5: Speed-Density Profile Dialog 56
Figure 4-6: Using a custom occupant profile 58
Figure 4-7: New Behavior dialog 58
Figure 4-8: Behavior Property Panel 58
Figure 4-9: Exit Chooser dialog 59
Figure 4-10: Behavior actions list 59
Figure 4-11: Example of action order for a behavior 60
Figure 4-12: Goto Waypoint creation panel 60
Figure 4-13: Goto Rooms creation panel 61
Figure 4-14: Goto Elevators creation panel 61
Figure 4-15: Wait creation panel 62
Figure 4-16: Adding occupants individually 62
Figure 4-17: Add Rectangular Group of Occupants Property Panel 63
Figure 4-18: Editing the distribution of profiles 63
Figure 4-19: Distributing occupants in a rectangular region 63
Figure 4-20: Distributing occupants through entire rooms 64
Figure 4-21: Edit Group Distributions dialog 65
Figure 5-1: Creating a View 66
Figure 5-2: Camera Tour property panel 68

Figure 5-3: Camera Tour View property panel 68
Figure 5-4: Tour preview playback panel 69
Figure 6-1: Property panel for the translate tool 71
Figure 6-2: Graphically moving an object 72
Figure 6-3: Creating an array of objects by using the translate tool 72
Figure 6-4: Property panel for the rotate tool 73
Figure 6-5: Rotating an object 73
Figure 6-6: Creating an array of objects using the rotate tool 74
Figure 6-7: Property panel for the mirror tool 74
Figure 6-8: Mirroring an object 75
Figure 6-9: Manipulator handles 75
Figure 6-10: Handle property panel 76
Figure 6-11: Door handles 76
Figure 6-12: Stair/ramp handles 77
Figure 6-13: Agent handles 77
Figure 6-14: Waypoint handles 78
Figure 7-1: Connectivity example 79
Figure 7-2: Select Connected Components dialog 80
Figure 7-3: Selecting an entire graph of connected components 81
Figure 8-1: The Time tab of the Simulation Parameters dialog 83
Figure 8-2: The Output tab of the Simulation Parameters dialog 83
Figure 8-3: The Paths tab of the Simulation Parameters dialog 84
Figure 8-4: SFPE Behavior tab of the Simulation Parameters dialog 85
Figure 8-5: The Steering Behavior tab of the Simulation Parameters dialog 86
Figure 8-6: The Run Simulation dialog showing a partially complete simulation. 87
Figure 9-1: Listing for an example summary report file. 89
Figure 9-2: A time history plot for door flow rates 91
Figure 9-3: 3D Results for the multi-floor-stairwell example problem 93
Figure 9-4: Floor location dialog for 3D results 95
Figure 9-5: Multi-floor layout options in 3D results (FDS model courtesy of Andreas Niggemeyer) 96
Figure 9-6: Movie option dialogs 99
Figure 9-7: FDS Slices 101
Figure 9-8: FDS preferences dialog 102
Figure 9-9: Occupant Contours 103
Figure 9-10: New Filtered Occupant Contour dialog 104
Figure 9-11: Occupant contours context menu 105
Figure 9-12: Occupant Contours Properties dialog 106
Figure 9-13: Effect of contour mesh resolution 106
Figure 9-14: Density contour properties dialog 107
Figure 9-15: Effect of influence radius on contours 108

1. Introduction

Pathfinder is an agent based egress and human movement simulator. It provides a graphical user interface for simulation design and execution as well as 2D and 3D visualization tools for results analysis.

Graphical User Interface

Pathfinder includes a graphical user interface that is used primarily to create and run simulation models. A screenshot of this user interface is shown in Figure 1-1. This screenshot displays a model of the Theater de Vest in Alkmaar, Netherlands. The model was created by Van Hooft Adviesburo. For clarity, the image shows only one of the dxf files used to create the geometry. The model includes 2177 occupants.

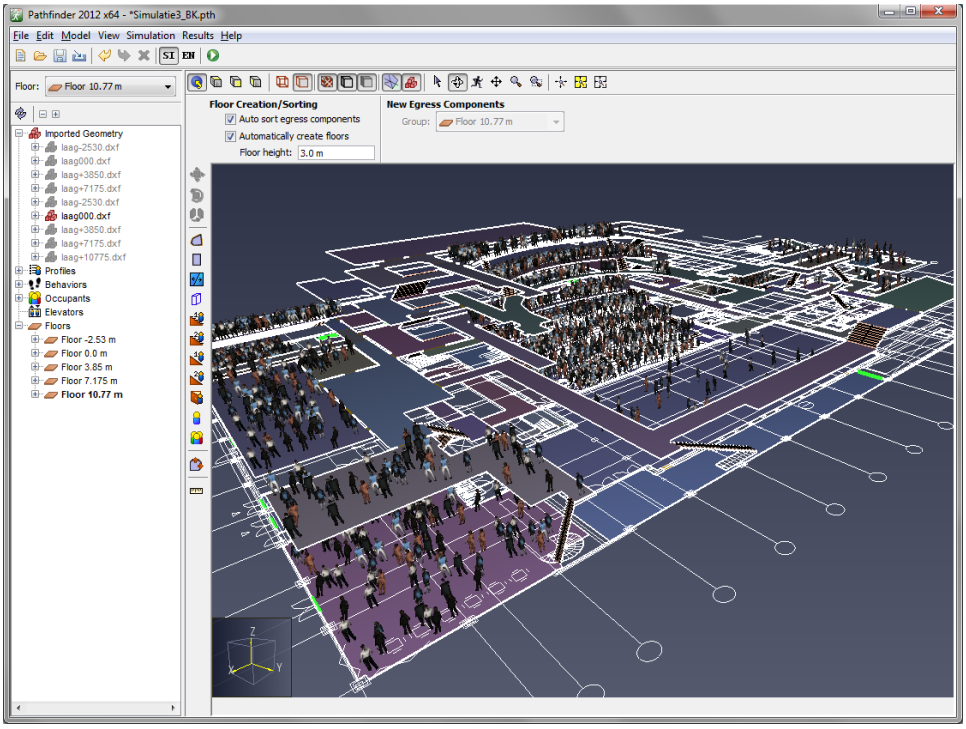


Figure 1-1: An example of the graphical user interface. Model of the Theater de Vest created by Van Hooft Adviesburo.

Pathfinder also includes a second program designed specifically for high-performance visualization of 3D time history. The 3D Results program is shown in Figure 1-2. In this image, occupants are gathering at a refuge area before proceeding to elevators. Transparency has been used to help view occupants on the refuge floor.

Pathfinder 2012 x64 - elevator tower refuge
File Agents Scene View Help
Floor Separation (m): 0 Set Wall Height (m): 0 Set
Exited: 0/700
Reverse
Stopped
Time: 1:24 / 10:03 Speed: 1 X
Framerate: 8.5 fps

Figure 1-2: An example of the 3D Results view, showing occupants gathering at a refuge area before proceeding to elevators. Transparency has been used to help view the occupants.

In addition to 3D visualization, Pathfinder also provides output in the form of 2D time history plots of CSV (comma separated values) out files and a text summary of room clearing times and doorway flow rates. An example time history plot can be seen in Figure 1-3. This plot shows the number of occupants in the refuge area and the total number of occupants in the building.

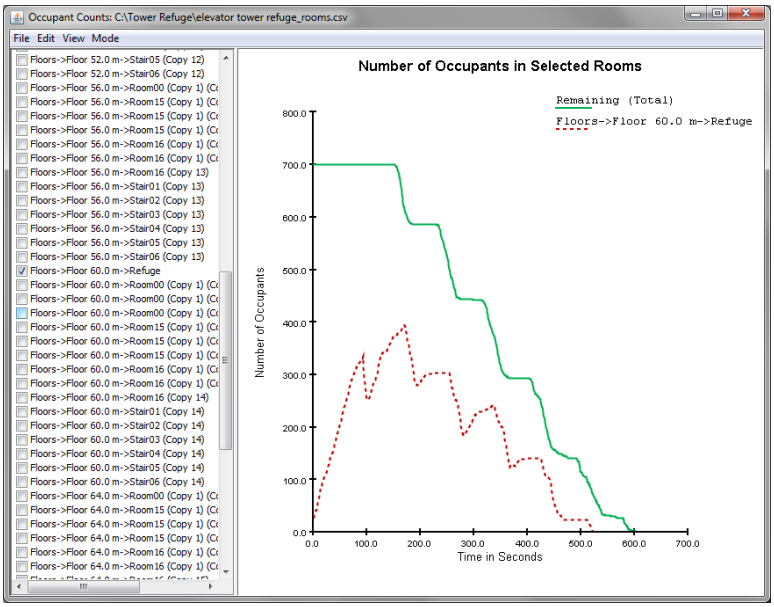


Figure 1-3: An example time history graph

Model Representation

The movement environment is a 3D triangulated mesh (Figure 1-4) designed to match the real dimensions of a building model. This movement mesh can be entered manually or automatically based on imported data (e.g. FDS geometry).

Walls and other impassable areas are represented as gaps in the navigation mesh. These objects are not actually passed along to the simulator, but are represented implicitly because occupants cannot move in places where no navigation mesh has been created.

Doors are represented as special navigation mesh edges. In all simulations, doors provide a mechanism for joining rooms and tracking occupant flow. Depending on the specific selection of simulation options, doors may also be used to explicitly control occupant flow.

Stairways are also represented as special navigation mesh edges and triangles. Occupant movement speed is reduced to a factor of their level travel speed based on the incline of the stairway. Each stairway implicitly defines two doors. These doors function just like any other door in the simulator but are controlled via the stairway editor in the user interface to ensure that no geometric errors result from a mismatch between stairways and the connecting doors.

Elevators are called to a floor when occupants arrive at the elevator door. The elevator model includes capacity, pick-up and discharge floors, and the ability to group elevators in banks.

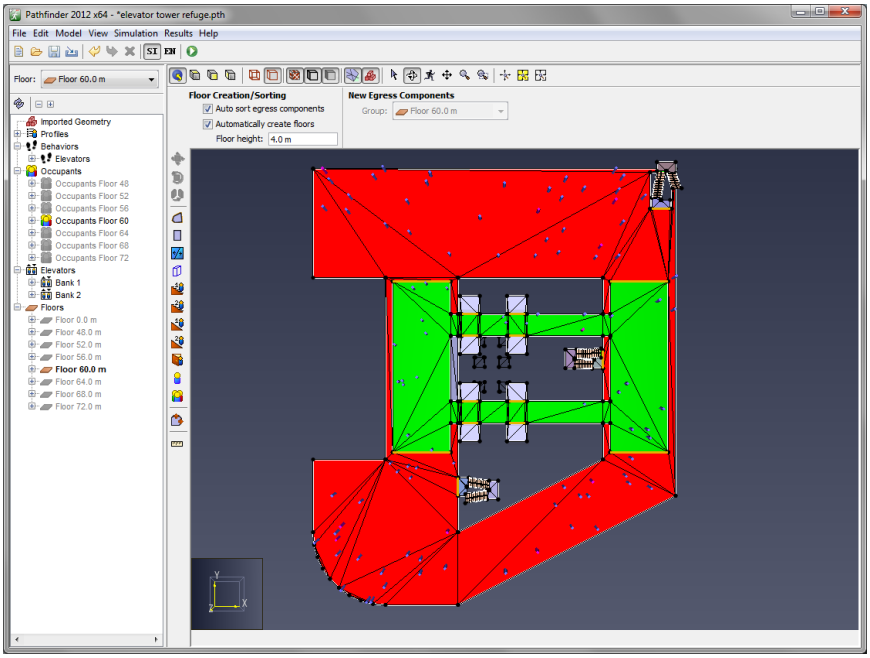


Figure 1-4: The triangulated navigation mesh used to represent one floor in the model.

Each occupant is defined by position, a profile that specifies size, speed, etc., and a behavior that defines goals for the occupant. The behavior allows scripting so that, for example, an occupant may wait at a location for a specified time and then proceed to an elevator. The occupant is represented as an upright

cylinder on the movement mesh and movement uses an agent-based technique called inverse steering. Each occupant calculates movements independently.

Simulation Modes

Pathfinder supports two movement simulation modes. In "Steering" mode, occupants use a steering system to move and interact with others. This mode tries to emulate human behavior and movement as much as possible. SFPE mode uses a set of assumptions and hand-calculations as defined in the *Engineering Guide to Human Behavior in Fire* (SFPE, 2003). In SFPE mode, occupants make no attempt to avoid one another and are allowed to interpenetrate, but doors impose a flow limit and velocity is controlled by density.

You can freely switch between the two modes within the Pathfinder user interface and compare answers. More information about both modes is provided in the Pathfinder Technical Reference (Pathfinder Resources).

Limitations and Known Issues

Pathfinder does not presently integrate results from a fire model or provide support for complex behaviors (e.g. family grouping).

Dynamic geometry is only partially supported (e.g. elevators, virtual escalators, and door opening/closing are supported, but trains and other moving surfaces are not).

Elevators are supported in evacuation-only circumstances. They do not model a general-purpose elevator system.

Simulator Name

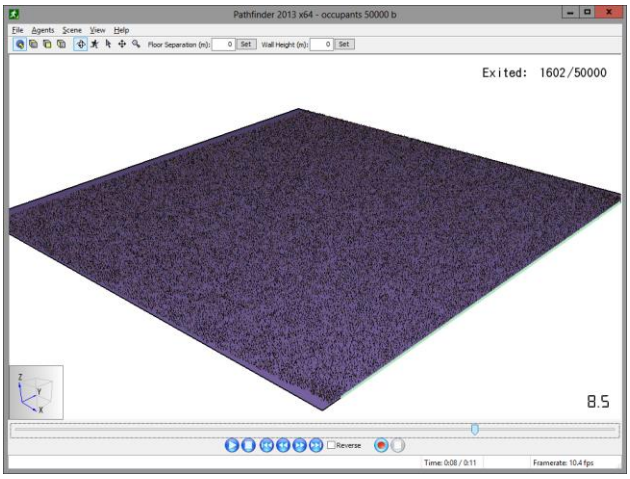
The name Pathfinder has been used previously to describe a 2D egress simulator created and used internally by Rolf Jensen and Associates. While the original Pathfinder inspired some of the features of the new simulator, the simulator described in this manual does not use any of the code from the original Pathfinder software.

System Requirements

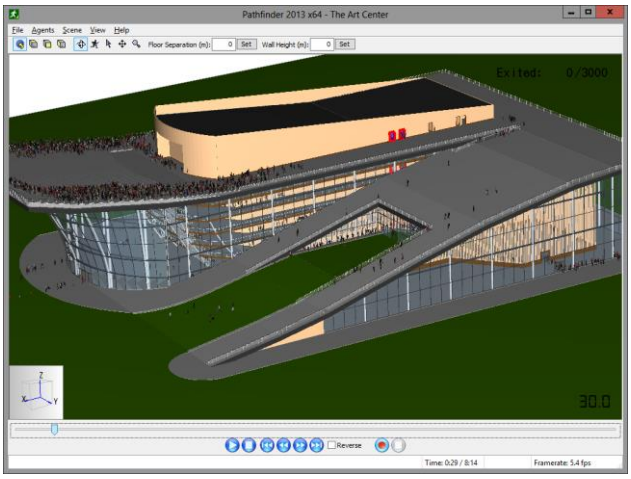
System requirements depend on the type of model being analyzed. To illustrate this, two different models were evaluated using a laptop running 64-bit Windows 8 Pro with an Intel Core i7 2.60 GHz processor, 8 GB of RAM, and NVIDIA NVS 5200M graphics card. The first model had a single room with 50,000 occupants and did not include any imported geometry. The second model imported a relatively complex Revit model to create the Pathfinder model and had 3,000 people, Figure 1-5.

Table 1-1 shows a comparison of the two models. The key parameters are the number of people in the model and the model complexity, measured by the number of navigation mesh triangles used in the Pathfinder solution and the number of imported Revit primitives (triangles). The simple model had only 4 triangles, with the consequence that the movement calculation of the path for each person is simple and that display performance is related to the drawing of people. The Revit model had 21,480 triangles for the navigation mesh but over 1,300,000 triangles for the Revit geometry.

The model with 50,000 people solved in about 18 minutes, while the Revit model with 3,000 people took about 5 minutes. The graphical display performance for the model with 50,000 people was responsive at 15 frames/sec, while the Revit model with the imported geometry displayed was slow at 5 frames/sec. When only the Pathfinder navigation mesh was displayed, the Revit model was responsive.



a. Model with 50,000 people



b. Revit import model with 3,000 people

Figure 1-5: Models used for system requirements comparison

Table 1-1: Comparison of performance for the two models

Parameter	Model	
	People	Revit Import
Number of occupants	50,000	3,000
Number navigation triangles	4	21,480
Number Revit face primitives	0	1,300,000
CPU solution time (s)	1090	297
Navigation mesh display rate (fps)	~15	~70
Imported geom display rate (fps)	n/a	~5

The minimum requirements to run Pathfinder include:

- 32 or 64-bit Windows XP (Windows 7 or higher strongly recommended)
- A processor the performance of an Intel i5
- 4 GB of RAM
- Graphics support for OpenGL 1.2

For a balanced performance we recommend:

- 64-bit Windows 7 or higher
- Intel i7-3770 (3.4 GHz, 4 Cores) processor
- 8 GB of RAM
- Graphics support for OpenGL 3.2 with an installed graphics card for large Revit models.

Revit models place a premium on graphics capability. We have found that mid-cost gaming graphics cards (GeForce GTX 570 / Radeon HD 7870 or equivalent) allow us to display relative large Revit models with good performance. Benchmarking sites that we find useful to compare CPU and graphics card performance can be found at: http://www.cpubenchmark.net/ and http://www.videocardbenchmark.net/.

Contact Us

Thunderhead Engineering
403 Poyntz Avenue, Suite B
Manhattan, KS 66502-6081
USA

Sales Information: sales@thunderheadeng.com
Product Support: support@thunderheadeng.com
Phone and Fax: +1.785.770.8511

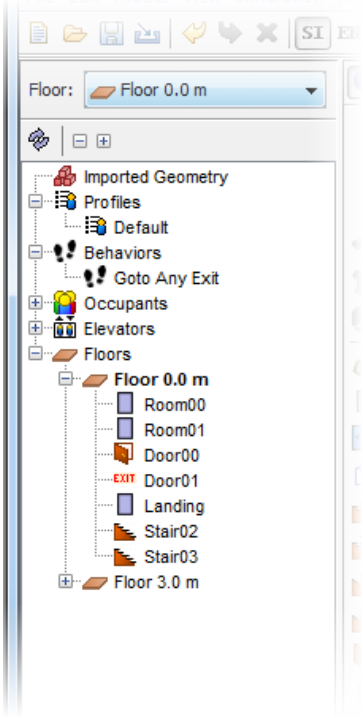
2. Pathfinder Basics

Pathfinder provides three main views for working on evacuation models: the 2D View, 3D View, and Navigation View. These views represent your current model. If an object is added, removed, or selected in one view, the other views will simultaneously reflect the change. Each view is briefly described below.

- **Navigation View**: This view lists all objects in the model in a hierarchical format. It can be used to quickly locate and modify objects by name.
- **3D View:** This view shows a 3D representation of the current model. The model can be explored and modified using various tools.
- **2D View:** This view is very similar to the 3D View, but it provides an additional snapping grid and an orthographic view of the model.

Navigation View

The Navigation View helps you quickly find objects and data that are not always easily accessible from the 3D and 2D views.



The Navigation View is arranged in six groups:

1. *The Imported geometry* group stores items that were imported from an image or an FDS, PyroSim, or DXF model. These objects do not affect the simulation but are carried through to help with results analysis. They can also be used to automatically extract rooms.
2. The *Profiles* group contains the occupant profiles that have been created using the **Edit Profiles** dialog.
3. The *Behaviors* group contains user-defined scripts that tell occupants how to behave.
4. The *Occupants* group contains every occupant in the model. If occupants are added to the model using a tool that adds more than one occupant at a time, they will be collected in a sub-group.
5. The *Elevators* group contains evacuation elevators in the model.
6. The *Floors* group defines the floors in the model, and each floor contains all geometry necessary to create a movement mesh, including room, stairway, ramp, door, and exit definitions.

The buttons directly above the Navigation View perform the following actions:

Auto Expand Selection when an object (or occupant) in the 3D or 2D view is selected, this action will expand the groups of the Navigation View as needed to show the selected object.

Collapse All collapses all expanded groups in the Navigation View.

Expand All expands all groups in the Navigation View (including sub-groups)

The **Floor** box above the view can be used to manage floors. Any time a room, stair, ramp, or door is created it is added to a floor group matching the current selection in the **Floor** box. Changing the selection in the **Floor** box will cause the newly selected floor to be shown and all other floors to be hidden. Also, the **Z** property for all drawing tools will automatically default to the height of the floor currently selected in the **Floor** box. The visibility of any object or group of objects can always be manually set using the right-click context menu. This technique is useful if you want to show two floors at the same time (e.g. when creating a stairway).

3D and 2D Views

The 3D and 2D views as shown in Figure 2-1 are the main views in which drawing is performed in Pathfinder. Both views contain tools to draw egress geometry and navigate in a model. The main difference between the two views is that the 3D view allows the model to be viewed from any direction, whereas the 2D view only allows viewing from one, orthographic direction. In addition, the 3D view contains no snap grid, whereas the 2D view does. The 3D view is entered by selecting the perspective camera, , and the 2D view is entered by selecting one of the orthographic cameras, , , or .

At the top of the view is several buttons that show different camera modes, display options, and navigation modes. The panel under this is known as the **property panel** and is a selection context-sensitive panel. If a drawing tool is selected, it will show properties that can used to help draw. If no drawing tool is selected, and an object or several objects are selected, this panel will show the properties relevant to the selection. The panel of buttons on the left shows move/copy and drawing tools. The small panel at the bottom displays messages relevant to the current tool.

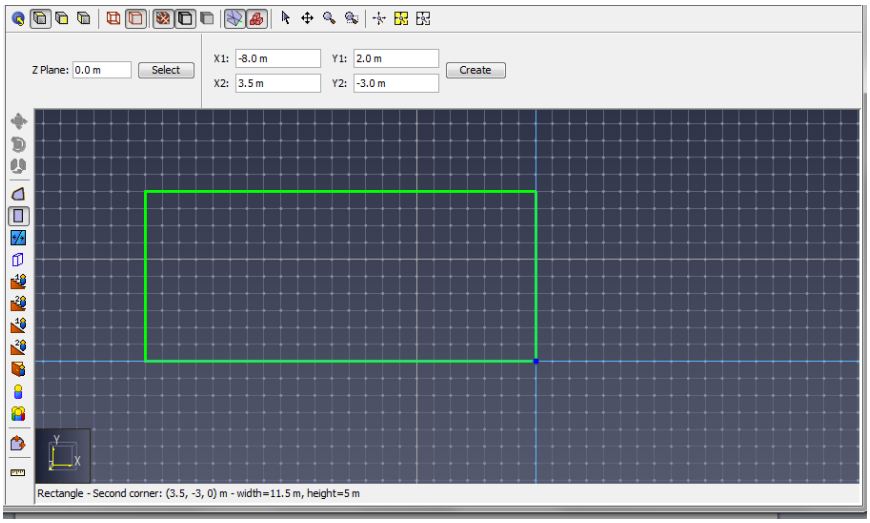


Figure 2-1: 3D and 2D views

Navigating the 3D view

Several tools are provided for navigating through the model in the 3D view, including orbit, roam, pan, and zoom tools.

The main navigation tool for the 3D view is the Orbit tool, . By left-clicking and dragging, the model is rotated about its center point. The scroll-wheel can be used to zoom in and out on a specific point. Holding SHIFT on the keyboard and then clicking and dragging will pan the camera, and holding ALT while dragging will zoom in and out.

Another navigation tool in the 3D view is the Roam tool, . This tool allows the camera to move in and out of the model at will. Without holding any keyboard keys, dragging the mouse will cause the camera to rotate about the camera's location. So dragging the mouse up will make the camera look up, and dragging it left and right makes it look left and right. Holding CTRL while dragging will make the camera move forward and backward in the XY plane, and holding ALT while dragging will make the camera move up and down along the Z axis. This tool has a higher learning curve but is the most flexible viewing tool because it allows the camera to be placed anywhere in the model.

The other navigation tools include a pan/drag tool, which moves the camera left and right and up and down, a zoom tool, which zooms in and out of the model while click-dragging, and a zoom box tool, which allows a box to be drawn that specifies the zoom extents.

Pathfinder can also be navigated while using the Selection/manipulation tool, . To Orbit the camera while in perspective view, use a right-click and drag combination. Similarly, use a middle-click and drag to Pan in perspective view.

Navigating the 2D view

Navigation in the 2D view is simpler than in the 3D view. The selection tool not only allows objects to be selected if single-clicked, but it allows the view to be panned by middle or right-clicking and dragging, and the view to be zoomed by using the scroll wheel. The drag and zoom tools are also separated into separate tools for convenience.

Resetting the view

At any time, the camera can be reset by pressing CTRL+R on the keyboard, or selecting Reset All tool, . This will cause the entire model to be visible in the current view. For all navigation tools but the Roam tool, reset will make the camera look down the negative Z axis at the model. For the roam tool, however, reset will make the camera look along the positive Y axis at the model.

The camera can also be reset to the current selection at any time by pressing CTRL+E. This will cause the camera to zoom in on the selected objects and the orbit tool to rotate about the center of their bounding sphere.

Filling the view

Very similar to resetting the camera, the view can be fit by pressing F on the keyboard or selecting the Fit View tool, . The difference between the Fill View and Reset All tools is that filling the screen does not change the view angle of the camera. Instead the camera will recenter/rezoom to fit the screen.

Drawing in the 3D and 2D views

Drawing can be performed in both the 3D view and the top 2D view. The 3D view allows the user to see the model from any angle, but most tools restrict drawing in the XY plane. The top view completely restricts drawing to the XY plane, but it also displays an optional snap grid. The snap grid size can be set under **Edit snap grid spacing** in the **View** menu, and it can be turned off by deselecting **Show Snap Grid** in the **View** menu.

Drawing is performed in one of the two following modes:

- **Normal Mode:** Single-click a drawing tool button on the left side of the view. Draw the object using the instructions in the appropriate section of the manual. When the object has been completed, the drawn object(s) will be selected and the view will revert to the previous navigation tool.
- **Sticky Mode:** Double-click a drawing tool's button on the left panel before beginning to draw. When the object is completed, the same drawing tool will remain selected and more objects can be drawn with the tool. To escape this mode, press ESC on the keyboard, and the previous navigation tool will be selected. A green dot on the tool's icon indicates that the tool is currently in sticky mode. Single-clicking the tool's icon again will turn off sticky mode but keep the tool selected.

At any time while drawing, the user can press escape, which causes the current object to be cancelled and the previous navigation tool to be selected.

For each tool there are often two ways to create its object. One way is to draw the object graphically using the mouse and keyboard. The other is to interactively create the object by typing information such as coordinates, widths, etc. in the tool's property panel. The property panel will update the graphical preview immediately to reflect changes in the input. This allows fine-grained control in creating the object. The individual drawing tools are discussed in Creating Movement Space.

View Options

Pathfinder provides a variety of view options for displaying both navigation geometry and imported geometry that can also aid with drawing. This includes options for rendering geometry, displaying agents, coloring rooms, and setting the transparency of rooms.

Render Options

In the toolbar above the properties window in the 2D and 3D views, there are a number of buttons as shown in Figure 2-2 that control how geometry is rendered.

Figure 2-2: Render options

From left to right, the buttons are **Wireframe Rendering**, **Solid Rendering**, **Show Materials**, **Show Object Outlines**, **Smooth Lighting**, **Show Navigation Geometry**, **Show Imported Geometry**, and **Show Occupants.**

- **Wireframe Rendering**: displays imported 3D geometry as wireframe only. This option is mutually exclusive with the solid rendering option. It is useful for drawing doors in the 2D view when 3D geometry has been imported.
- **Solid Rendering**: displays imported 3D geometry filled in. This is selected by default.
- **Show Materials**: shows materials applied to the faces of imported 3D geometry. This is selected by default.
- **Show Object Outlines**: shows the outlines of 3D imported geometry. This is similar to showing the wireframe and solid versions at the same time.
- **Smooth Lighting**: uses a more realistic shading model to show all geometry. This may be a little slower to render on older graphics cards.
- **Show Navigation Geometry**: This toggles the visibility of all the navigation geometry. It does not affect anything else (including imported geometry and occupants).
- **Show Imported Geometry**: This toggles the visibility of all imported 3D geometry.
- **Show Occupants**: This toggles the visibility of occupants.

Occupant Display

Occupants can be displayed using a number of options. They can be viewed as simple shapes, including disks and cylinders. They can also be displayed as the artist's mannequin or as their respective human avatars specified in their profiles. These options are available under **View** menu and **Agents** submenu.

Coloring Rooms

Rooms can be colored in a variety of ways. All coloring options are available under the **View** menu and **Color Rooms** submenu. The default option is to display each room with a unique color. They can also be colored by occupant density, with red indicating high density and blue low density. The last option is to use a mixed mode. In this mode, the rooms are only colored by concentration if they contain agents; otherwise, they are colored with their unique colors.

Room Opacity

Sometimes it is useful to be able to see through rooms and stairways, such as when drawing on top of an imported background image. To change the opacity of a set of components, select them and in the property panel, change the opacity. Opacity settings will carry through to 3D results visualization.

Model Organization with Groups

The main method of organization in Pathfinder is to use groups. In every model there are already some implicit groups that cannot be modified, including **Imported Geometry**, **Profiles**, **Behaviors**, **Occupants**, **Elevators**, and **Floors** as shown in Figure 2-3. Sub-groups can be created to further organize the model as discussed in the following sections.

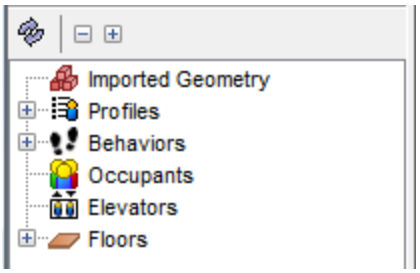


Figure 2-3: Predefined groups

Creating sub-groups

Sub-groups can be created under **Imported Geometry**, **Occupants**, **Elevators,** and **Floors** (floors are discussed in the section, Floors). Groups can also be created in other sub-groups. To create a new group, right click the desired parent group in the navigation view and select **New Group…** or select **New Group…** from the **Model** menu. A dialog will display allowing the user to select the parent group (which will automatically be selected if performed from the right-click menu) and a name for the new group. Click "OK" to create the new group.

Changing groups

An object can be moved from one group to another at any time. To change an object's group, drag the object to the desired group in the Navigation View or right click the object and select **Change Group…**. This will show a dialog that will allow the user to choose the new group. The options shown for the new group will only be valid groups for which the group can be changed. Select "OK" to change the group.

3. Creating Movement Space

Pathfinder is built on the idea of creating floor space on which occupants can walk. Every navigation component drawn in Pathfinder is some piece of flooring that can be travelled on, which can range from floors, to doorways, to stairs. Obstructions exist as holes in the floor.

The main egress components include rooms, which are empty floor spaces bounded by walls, doors, which connect rooms on the same level, stairs/ramps, which connect rooms on different levels, and elevators, which connect multiple levels. Rooms can have any polygonal shape, and can never overlap on the same level. Doors can be either thick if they are occupying a doorway (the area between two rooms) or thin if they are simply connecting two touching rooms. Stairs/ramps are always rectangular and implicitly contain a thin door on each end to connect the adjacent rooms. Elevators can be any shape and can travel in any direction.

To organize egress components, Pathfinder provides the concept of floors, which group together components at different Z locations.

Floors

Floors are the primary method of organization in Pathfinder. At their most basic level, they are simply groups in which rooms, doors, stairs, ramps, and exits can be placed, but they also control the drawing plane for most tools and filtering of imported geometry.

In every Pathfinder model, at least one floor must exist, and at any given time, there is one active floor. Whenever any navigation object is drawn, it will either be placed in the active floor or a subgroup of the active floor.

By default when a new model is started, there is one floor at Z=0, and additional floors are either created automatically depending on where the geometry is drawn or manually created. In addition, new navigation components are automatically sorted into the appropriate floor when drawn.

Automatically creating floors

When nothing is selected in the model, the **Floor Creation** panel is shown, as in Figure 3-1. This panel controls the automatic creation of floors and automatic sorting of new objects into floors.

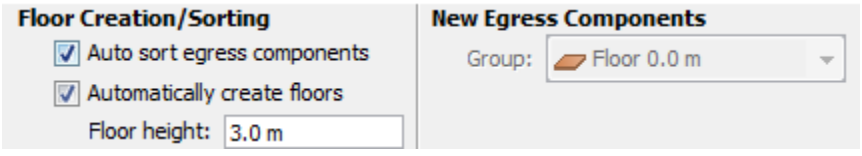


Figure 3-1: Floor Creation panel

- **Auto sort egress components** – If this is checked, navigation components are automatically sorted into the appropriate floor when created or modified; if this is unchecked, new navigation

components are placed in the group specified under **New Egress Components** and remain in this group until manually moved.

- **Automatically create floors** – If this is checked, floors are automatically created as navigation components are created and modified.
- **Floor height** – This specifies the height at which new floors are automatically created. If a navigation component is created or moved to a location that is at least this distance from the previous floor, a new floor will be created at a multiple of this distance from the previous floor.
- **Group** – If **Auto sort egress components** is unchecked, this drop-down specifies the group/floor for new navigation components.

The following scenario demonstrates how objects are organized when auto-sort and auto-floor-creation are enabled (organization of the model is shown in Figure 3-2):

1) A new model is created. The floor height is left at the default of 3 m.
2) "Room00" is drawn at Z=0 m, and is auto-placed in "Floor 0.0 m."
3) "Room01" is drawn at Z=1.5 m, and is auto-placed in "Floor 0.0 m."
4) "Stair01" is drawn connecting "Room00" to "Room01" and is auto-placed in "Floor 0.0 m."
5) "Room02" is drawn at Z=-1.5 m. A new floor, "Floor -3.0 m" is auto-created, and "Room02" is auto-placed in it.
6) "Stair02" is drawn connecting "Room02" and "Room00" and is auto-placed in "Floor -3.0 m."
7) "Room03" is drawn at Z=7.5 m. A new floor, "Floor 6.0 m" is auto-created, and "Room03" is auto-placed in it.

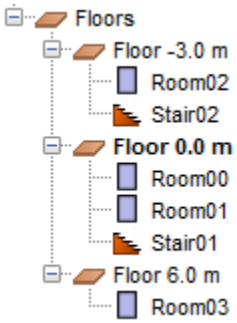


Figure 3-2: Auto floor creation and sorting

In this example, only rooms and stairs were created. The floors were automatically created and the rooms and stairs were automatically sorted into the appropriate ones.

Using Auto-sort with an Existing Model

Automatic floors can be created and components sorted into the floors by performing the following:

1) Open the model.
2) Clear the selection so that the **Floor Creation** panel is visible (Figure 3-1).
3) Ensure that the desired creation/sorting options are enabled and that the correct floor height for the model is set.

4) Select all the components that should be auto-sorted (if everything should be sorted, select the **Floors** top node).
5) Right-click the selection, and from the shortcut menu, click **Sort into Floors**.
6) The appropriate floors will be created and all selected items will be sorted into the appropriate floors.

NOTE: This will not delete any existing floors. If there are undesired existing floors, first move the navigation components out of them to another floor that will be kept, delete the undesired floors, and then perform **Sort into Floors**.

Manually creating floors

Floors can also be created manually at any time. To do so, click on the floor drop-down box above the Navigation View, and select **<Add New…>** as shown in Figure 3-3. A dialog will open asking for the floor location. Enter the Z plane location or click on a snap point in the 3D or 2D view and click okay. This Z plane will be used to update the drawing tools' working Z location when this floor is made active. By default, the name of the floor is "Floor x" where *x* is the working plane of the floor. If **Set as active floor** is selected in the new floor dialog, the floor is set to be the active floor in the model after being created. If **Resort existing egress components into new floor** is selected, then all existing components that belong in the new floor will be moved to it.

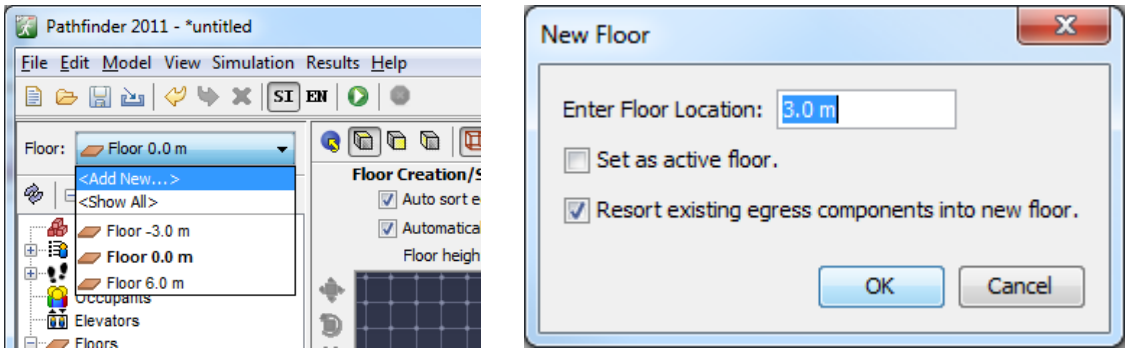


Figure 3-3: Adding a new floor

Changing the active floor

To change the active floor, click the floor drop-down box as shown in Figure 3-3, and select the desired floor. This will make that floor active and all other floors non-active.

Whenever the active floor is changed, the following additional changes take place in the model:

- The floor, all objects in the floor group, and all occupants on the floor are set visible.
- All other floors, sub-objects of other floors, and occupants on those floors are set hidden.
- The working plane of the room and wall subtraction tools is set to the working plane of the floor.
- A clipping filter is applied to imported geometry so that only geometry within the Z clipping planes of the active floor is visible.

Showing all floors

To show all floors click the floor drop-down box as shown in Figure 3-3, and click **<Show All>**. This will additionally show all occupants on the floors and all sub-objects of the floors groups, and it will set the import filter to the union of all the floors' filters.

Floor Properties

To edit a floor's properties, first select the desired floor. The property panel as shown in Figure 3-4 will appear, showing the floor's name, its working Z location, and the Z clipping planes for imported 3D geometry. It also shows some statistics of the floor including the total area of the floor (**Area**), number of people on the floor (**Pers**) and density of people.

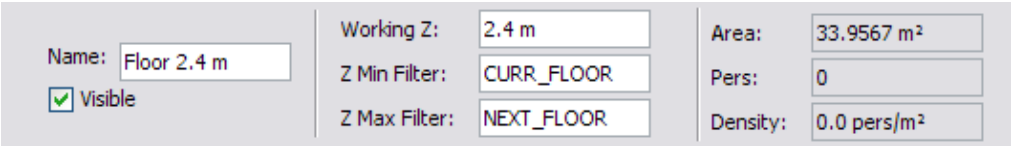


Figure 3-4: Floor property panel

The **Working Z** property controls the plane on which new rooms and wall obstructions are drawn.

The Z min and max filters control the clipping planes of imported 3D geometry when the floor is visible. Anything below Z min and above Z max is clipped. The **Z Min Filter** property can either be a Z plane location or can have the special value, **CURR_FLOOR**. If it is **CURR_FLOOR**, then the clipping plane is set to the working Z location if there are any floors below this floor or $-\infty$ if there are no floors below. The **Z Max Filter** can be a Z plane location or can have the special value, **NEXT_FLOOR**. If it is **NEXT_FLOOR**, then the Z plane is set to the working Z plane of the next higher floor if one exists, or $+\infty$ if there are no higher floors.

Rooms

Rooms are open space on which occupants can freely travel. Each room is bounded on all sides by walls. Rooms can be drawn so that they touch each other, but an occupant can only travel between them if they are connected by a door. Only one room can occupy a given space at any time, so if one room is drawn overlapping another, the overlapping area will be subtracted from the old room and given to the new. Rooms can also be merged into one, separated into constituent parts, and have internal, thin boundaries drawn in them. These features are discussed in the following sections.

Adding new rooms

Pathfinder provides two tools for adding new room geometry:

- **Polygonal Room Tool:** The Polygonal Room tool ( ) allows for the creation of complex shapes with any number of vertices. Left-click anywhere in the model to set the first point, and continue left clicking to add more points to the polygon. When at least three points are defined, right-clicking will close the polygon and complete the shape. Alternatively, x-y coordinates can be entered from the keyboard with the **Add Point** and **Close Polygon** buttons from the property panel.

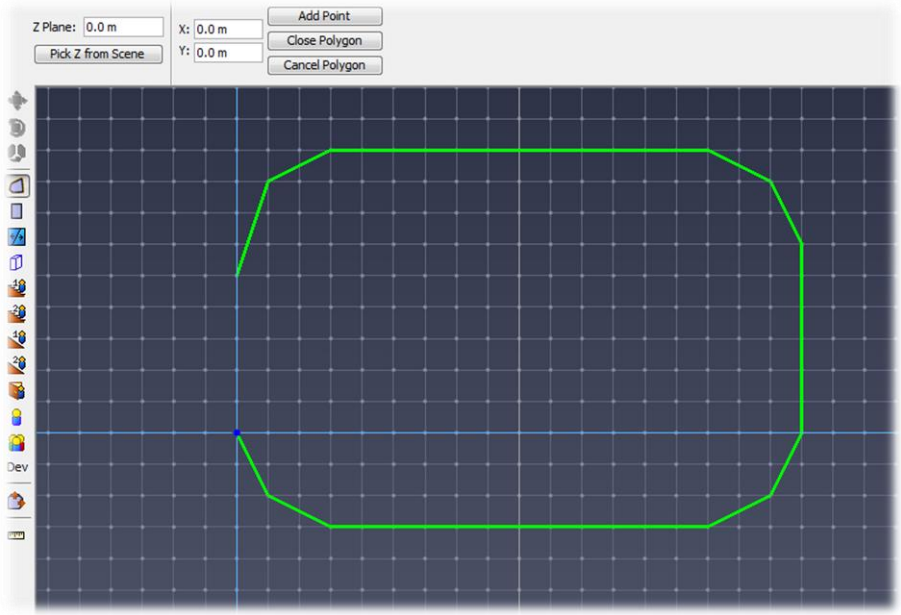


Figure 3-5: Drawing a room with the polygon tool

- **Rectangular Room Tool:** The Rectangular Room tool ( ) creates simple rectangular geometry by left-clicking two points in the model. The rectangular area can also be created by entering coordinates for two points in the property panel and clicking the **Create** button.

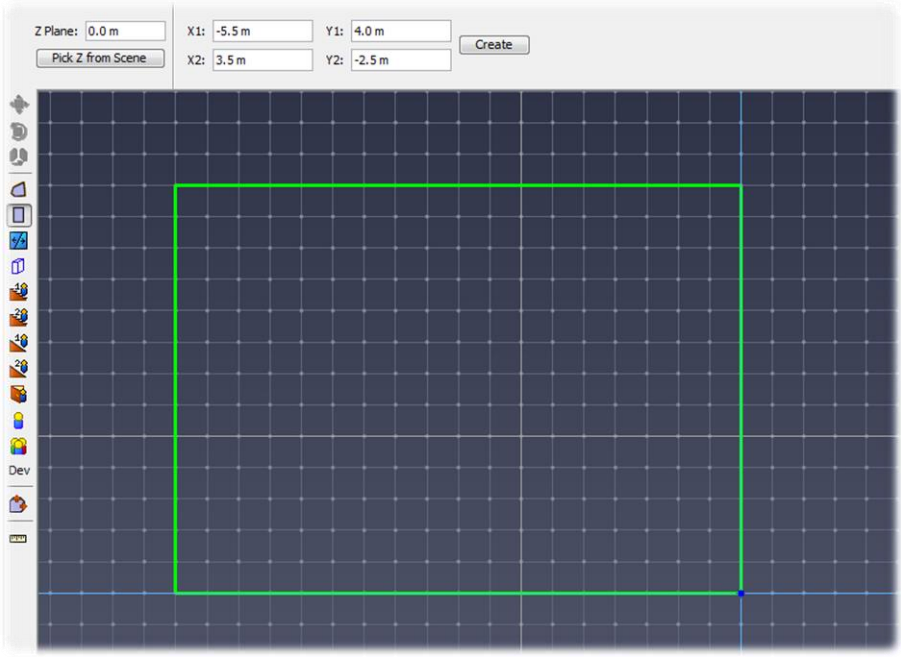


Figure 3-6: Drawing a room with the rectangle tool

In addition to creating new areas, both of these tools can be used on existing geometry to create negative areas. Creating new geometry over existing areas removes any interfering portion from those

areas. The newly created geometry can then be deleted, leaving the negative space behind. This is discussed further in the section, Arbitrarily-shaped obstructions (desks, tables, etc.).

Drawing Plane

Each room tool draws on a particular Z plane that is specified in the property panel for the tool as shown in Figure 3-7.

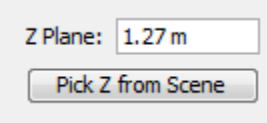


Figure 3-7: Drawing plane property

The Z plane can be specified either manually by typing the location into the **Z Plane** field or by picking the location from the **2D** or **3D View** as follows:

1. Select one of the room drawing tools.
2. In the tool property panel, click **Pick Z from Scene**. This will clear the tool property panel while waiting for the user to click a location.
3. Click a point in either the **3D View** or a **2D View**. The tool property panel will return for the selected drawing tool, and the **Z Plane** field will be filled with the Z coordinate of the clicked location.

Thin walls

Thin, internal walls or boundaries can be added to rooms with the Thin Wall tool ( ). To use this tool, click two points in the model as shown in Figure 3-8. Pathfinder will attempt to connect these two points with an internal boundary edge. NOTE: In some circumstances, Pathfinder may have trouble connecting the two points. If this occurs, try limiting the two points to one room or to the number of room boundaries crossed.

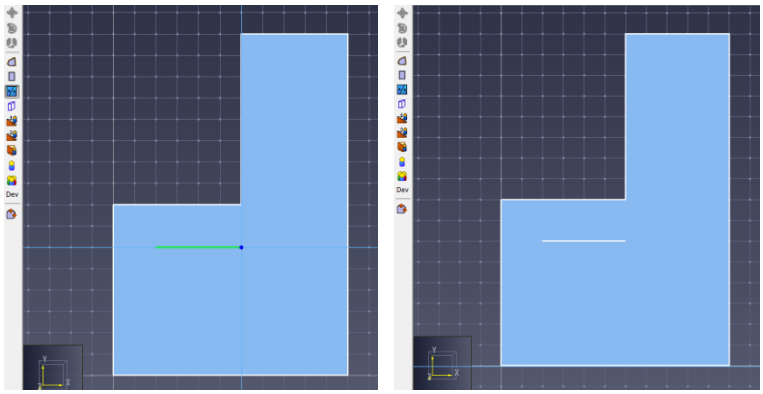


Figure 3-8: Adding a thin wall to a room

Splitting rooms

Rooms can be split into two or more pieces using the Thin Wall tool. To do this, specify the two points such that they are on the outermost boundary of the room to be divided. The original geometry will be divided into two or more new rooms with that line as the boundary between them as shown in Figure 3-9.

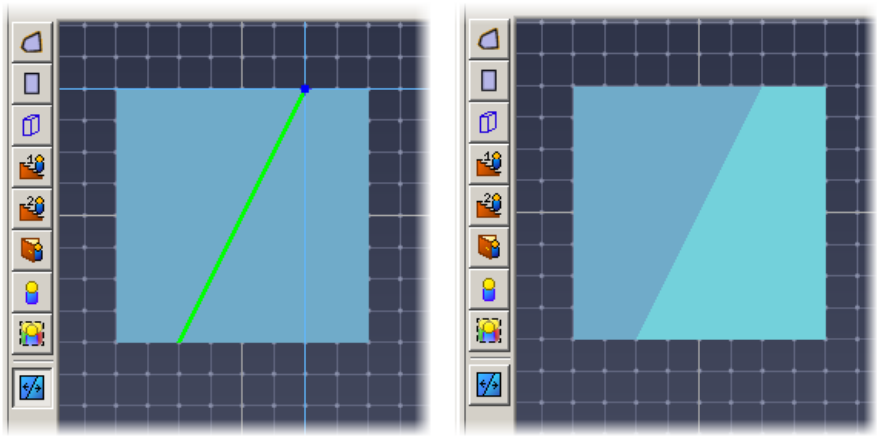


Figure 3-9: Dividing a room

Note that drawing a thin wall between two boundaries of the room will not always split the rooms into multiple pieces as shown in Figure 3-10. In this case, the tool will simply add a thin wall.

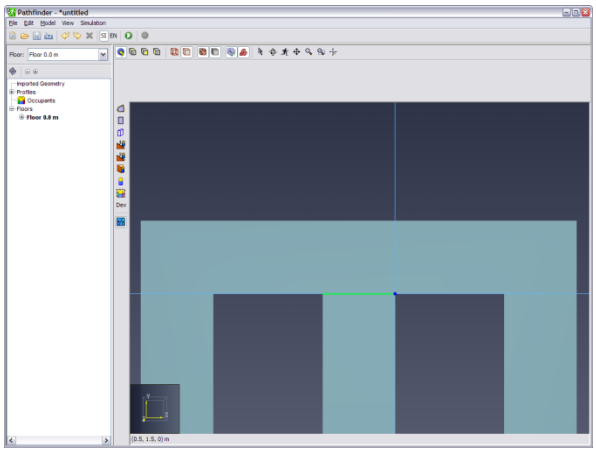


Figure 3-10: Special case for drawing a thin wall

Separating and merging rooms

In addition to dividing rooms, Pathfinder has two additional means to aid in creating more complex room geometry.

- **Merge:** The Merge command is used to join two or more rooms that share boundaries into one room. To use it, select the neighboring rooms and select **Merge** from either the **Model** menu or the right-click menu. NOTE: Rooms can be merged even if they do not lie in the same plane as

long as they share a common edge. They can also be merged with stairs and ramps, but the stairs and ramps will be converted into rooms and will lose their stair/ramp properties.

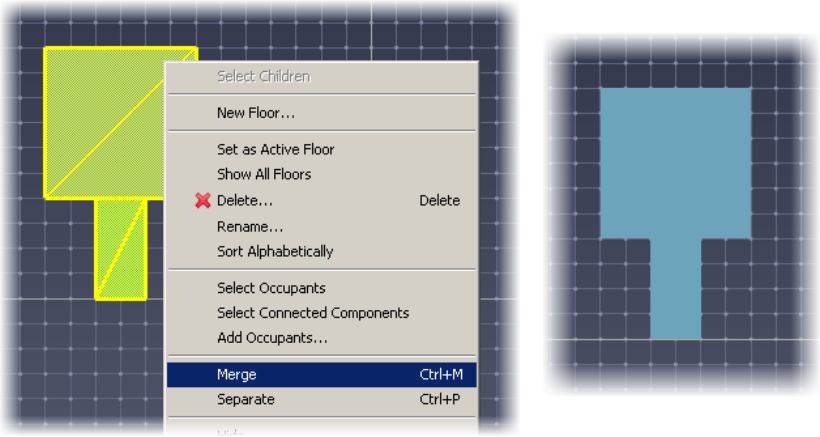


Figure 3-11: Merging rooms

- **Separate:** The Separate command breaks a room into its constituent parts along any negative space that divides it. To use it, select the room to be separated and select **Separate** from either the **Model** menu or the right-click menu.

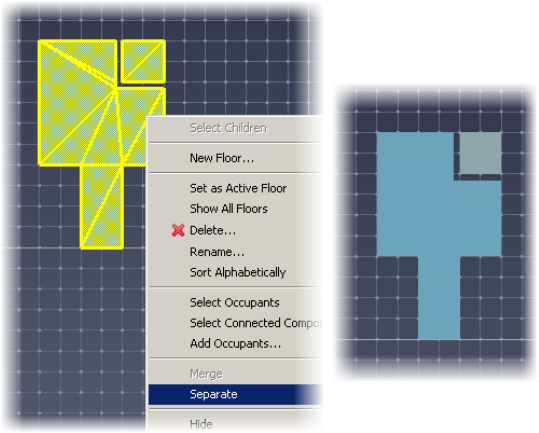


Figure 3-12: Separating a room

Room properties

To view and edit room properties, select a room. Its properties will be displayed in the property panel as shown in Figure 3-13.

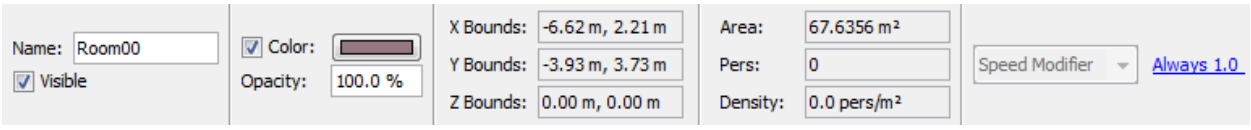


Figure 3-13: Room properties panel

- **Name**: An identifying name for room.
- **Visible**: Whether the room is currently visible. Uncheck to hide the room.
- **Color**: The color of the room. If unchecked, a default room color is used.
- **Opacity**: The opacity of the room. Making this value less than 100% will cause objects behind the room to become visible.
- **X**, **Y**, and **Z Bounds**: The geometric bounds of the room.
- **Area**: The geometric area of the room.
- **Pers**: The number of occupants currently placed in the room.
- **Density**: The number of occupants/area
- **Speed Modifier**: A time-variable factor that affects the speed of each occupant who travels in the room. When occupants travel in the room, their maximum speed is multiplied by this factor. This could be used, for instance, to represent the effect of smoke on occupant speeds.
  By default, the modifier is 1.0 throughout the simulation. To change this, click the link. The **Edit Speed Modifier** dialog appears as shown in Figure 3-14. In this figure, for instance, the room initially leaves occupant speeds unmodified, but over the first 40 seconds ramps occupant speeds down to 25% of their maximum values.

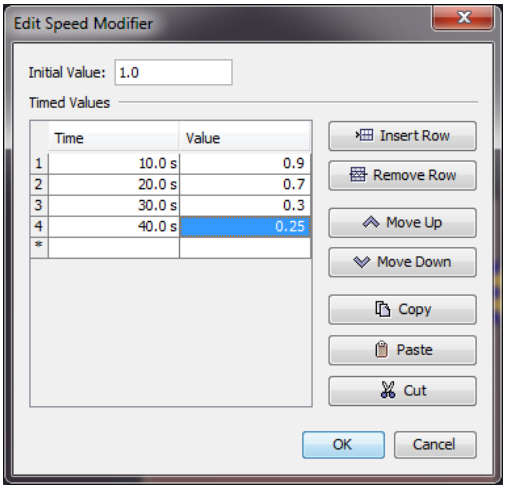


Figure 3-14: Edit Speed Modifier dialog

Preventing room-crossing

In some cases, such as modeling seating rows or shops in a mall, it may be desirable to only allow occupants to exit the room and not cross through it. This can be accomplished by making all the doors connected to the room one-way (see Doors) and ensuring that their directions point out of the room. Pathfinder provides a tool to make this easy. Instead of individually setting the one-way status of all the connecting doors, perform the following:

1. Select the room(s) that should not be crossable.
2. Right-click one of the rooms, and from the menu select **Make Doors Oneway…**.
3. A dialog will appear as in Figure 3-15. From this dialog, choose whether occupants can only **Enter** or **Exit** the room. If any of the room's doors were already marked as one-way, another

option will be provided to overwrite the one-way status of those doors. Press **OK** in this dialog to make the doors one-way.

Pathfinder will automatically calculate the correct directions for the doors to make the rooms exit-only or enter-only.

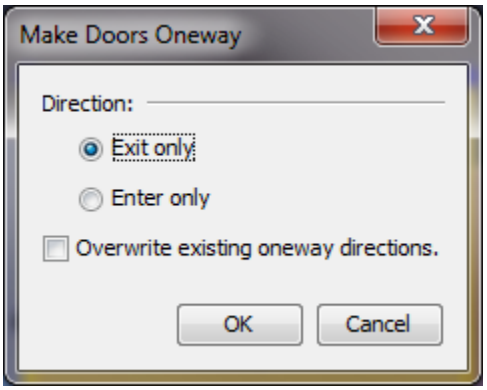


Figure 3-15: Make Doors Oneway dialog

NOTE: If any doors were shared between rooms in the selection, those doors will not be affected, as their direction would be ambiguous.

Obstructions/Holes

In Pathfinder, obstructions are modeled as holes in the navigation geometry. Holes can be created with an arbitrary polygonal shape or as thick walls.

Arbitrarily-shaped obstructions (desks, tables, etc.)

To model an obstruction (e.g. an office desk or other standing obstacle) within a room, the subtractive property of rooms is used. This means that the room containing the obstruction must already exist. To create the obstruction, select the **Add a Polygonal Room** tool or the **Add a Rectangular Room** tool and draw the shape and location of the obstructed area. This will subtract the area from the old room and create a new room. Next, delete the new room. A hole is left in the old room in the shape of the obstruction. This process is shown in Figure 3-16.

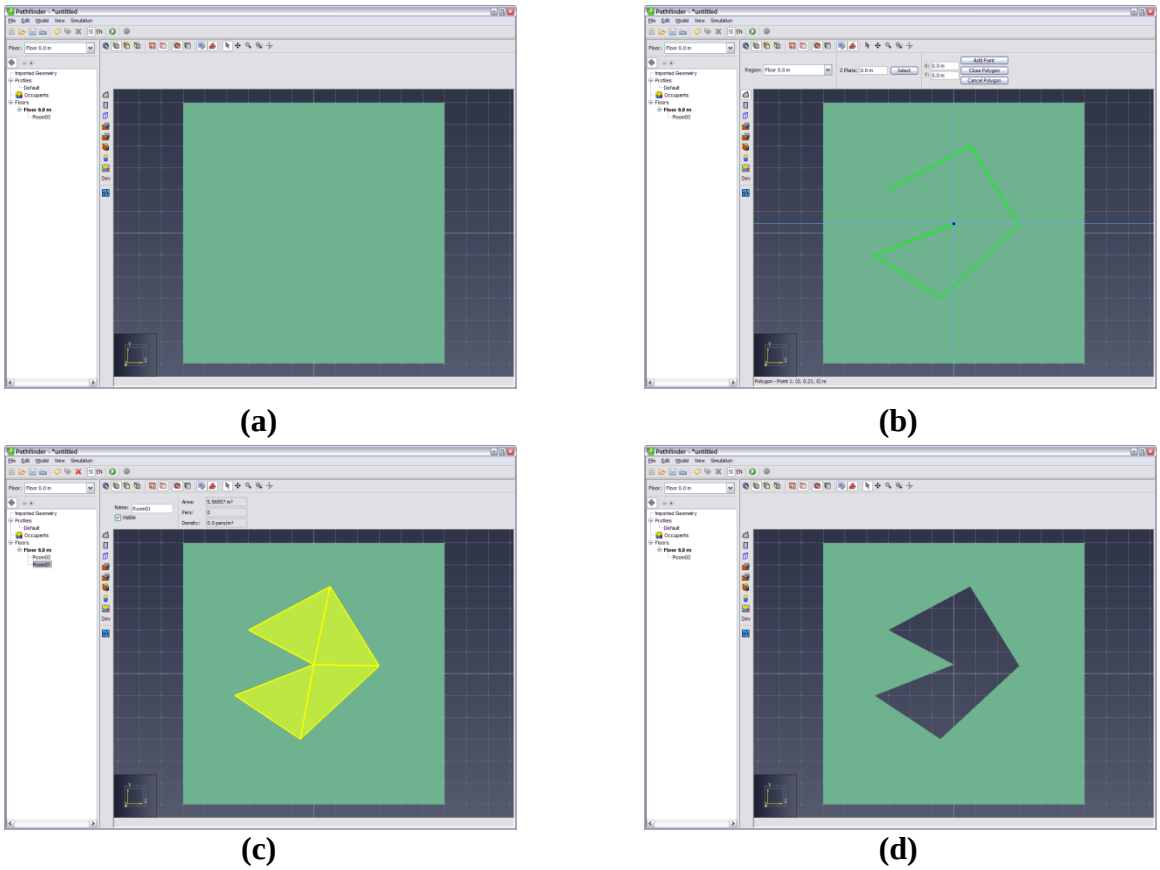


Figure 3-16: Creating an obstruction

Thick walls

The wall tool ( ) is used to make rectangular obstructions in existing geometry. To use this tool, enter the desired wall width in the property panel and click or click-drag the two points the wall is to pass through. Holding the shift key will switch between alignment left and right of the defining line. This tool is shown in Figure 3-17.

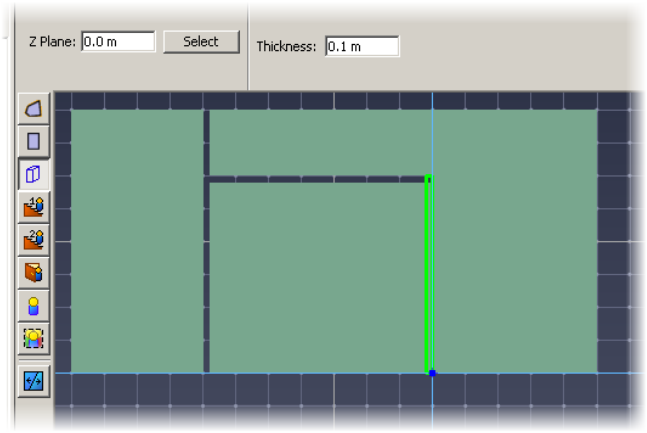


Figure 3-17: Subtracting walls

NOTE: The thick wall tool is not as robust as the thin wall tool. It operates by subtracting area from any room that lies in its specified Z plane, whereas the thin wall tool will follow the slope of multiple rooms to connect two points.

Doors

In Pathfinder, occupants cannot pass between two rooms unless they are joined by a door. Also, the simulator requires that each occupant must have a path to at least one exit door. Doors provide useful flow measurements in simulation results. Also, in the SFPE mode doors act as the primary flow control mechanism. You can add doors using the **Add a New Door** tool.

When adding doors, different parameters provide hints to Pathfinder for finding a valid door as shown in Figure 3-18. The **Max Width** parameter is used as a target width for the door. If the full width isn't available, Pathfinder will display a shorter door as you hover with the mouse. The **Max Depth** parameter represents the depth of the door cavity and is used to determine how far apart two rooms can be and still be joined by a door. Doors can be added between rooms that are separated by up to this distance.

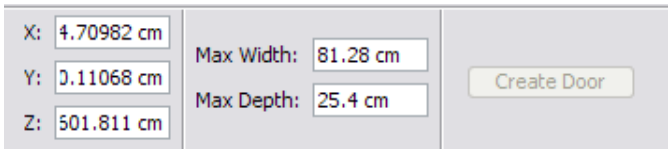


Figure 3-18: Door tool property panel

Thin doors

Thin doors can be used to connect two rooms that touch one another as shown in Figure 3-19. A door is needed in this example to allow occupants to travel from one room to the other. To create a door in this manner, first select the door tool and then use one of the following three methods:

- **Manual Entry:** Enter the coordinates of the door in the property panel. If the coordinates specify a valid door location, the **Create Door** button will enable. Click this button, and a door no larger than **Max Width** will be created. For thin doors, **Max Depth** will be ignored.
- **Single Click:** Move the cursor over the desired door location in the 3D or 2D view. A preview door will be displayed if the cursor is on a valid edge. The door displayed will lie either to the left or right of the hover point relative to the boundary edge, depending on whether **Max Width** is positive or negative. Single click to place the door. The previewed door will then be added to the model.
- **Click-drag:** Move the cursor over the location of one end point of the door, and click drag along the same edge. While dragging, a preview door will be displayed from the first to the second point. When the mouse is released, a door is created along the edge between the two specified points. Creating a door in this manner ignores all the properties in the tool panel.

The created door will appear as a thin, orange line in the 3D and 2D views as shown in Figure 3-19.

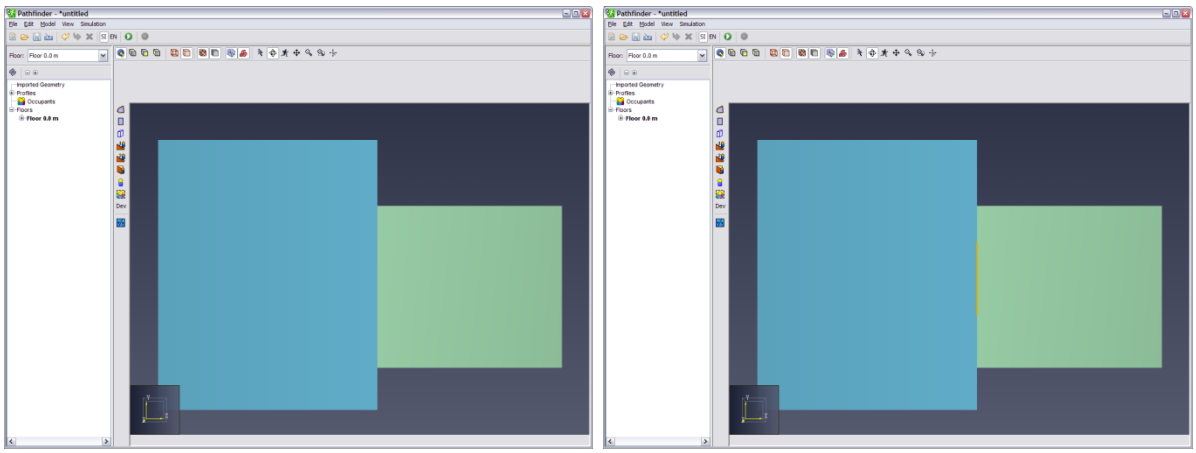


Figure 3-19: Adding a thin door to connect two touching rooms

Thick doors

Thick doors are often useful in realistic models, especially when CAD geometry has been imported. In real scenarios, rooms will not touch each other by infinitely thin walls as shown in Figure 3-20. To create a thick door to connect these rooms, first select the door tool and then use one of the following three methods:

- **Manual Entry:** Make sure **Max Depth** is greater than or equal to the distance between the edges the door will lie between. Enter a point on one of the edges in the property panel. If the coordinates specify a valid door location, the **Create Door** button will enable. Click this button, and a door no larger than **Max Width** will be created.
- **Single Click:** Make sure **Max Depth** is greater than or equal to the distance between the edges the door will lie between. Then move the cursor over the desired door location in the 3D or 2D view. A preview door will be displayed extending between the rooms if the cursor is on a valid edge. The door displayed will lie either to the left or right of the hover point relative to the boundary edge, depending on whether **Max Width** is positive or negative. Single click to place the door.
- **Click-drag:** Move the cursor over the edge of one of the rooms, and click drag along the corresponding edge on the second room. While dragging, a preview door will be displayed connecting the two edges. When the mouse is released, a door is created between the edges of the two rooms, where the diagonal of the rectangular door connects the two specified points. Creating a door in this manner ignores all the properties in the tool panel.

The created door will appear as an orange rectangle in the 3D and 2D views as shown in Figure 3-20.

When simulating, thick doors have a special representation: the area of the door will be partitioned in two, and each half is attached to its touching room. A thin door is placed in the middle of the area to represent the thick door. Note that the extra area attached to each room is neglected when the area of the room is reported in its property panel, but it is included during simulation.

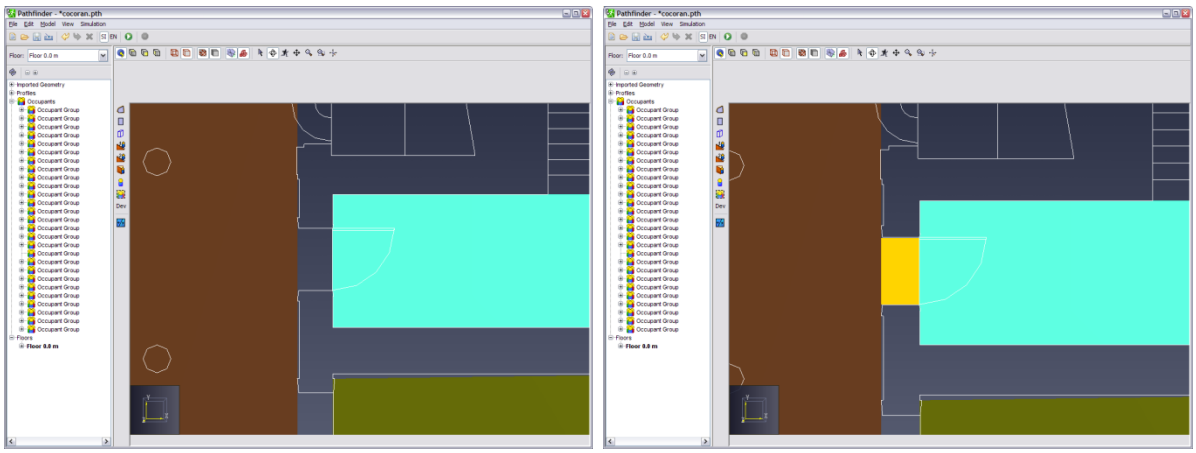


Figure 3-20: Adding a thick door in the empty space between two rooms

Door properties

To edit a door's properties, select the door. Its properties will appear in the property panel as shown in Figure 3-21.

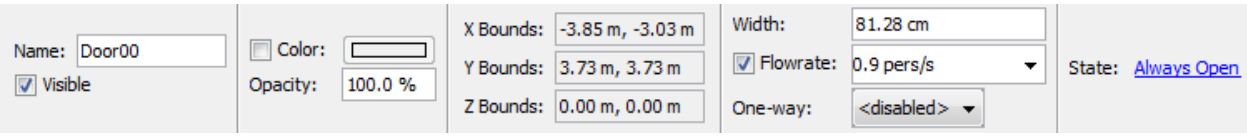


Figure 3-21: Door property panel

- **Width**: The width of the door. Changing this value will change the width of the door, but the value cannot exceed the length of its room edge.
- **Flowrate**: Checking this box overrides the default door flowrate setting in the **Simulation Parameters Dialog** (see Parameters on page 83). Setting this value controls the maximum occupant flowrate for the door in units of pers/t. This could be used, for instance, to specify a gated mechanism, such as a turnstyle. A value of .9 pers/s, for instance, would mean that one occupant can go through the door every 1.1 seconds (1/.9).
- **One-way**: The one-way direction of the door. A One-way door is one in which occupants can only travel through it in one direction.[1]
  NOTE: Even though a door can only be travelled through in two possible directions, the drop-down box allows **+X**, **-X**, **+Y**, and **-Y**. When one of these directions is chosen, the actual direction Pathfinder chooses is the closest along the door's normal.
- **State**: Indicates the timed opening and closing of the door. By default, all doors are always open throughout the simulation. To change this, click the link. The **Edit Door State** dialog will appear as shown in Figure 3-22. This dialog allows the initial state of the door to be specified as well as additional timed states. As shown in the figure, for example, the door is initially open, closes at t=10 s and then opens again at t=30 s.

[1] Occupants can ignore the one-way setting of doors if their profile has **Ignore Oneway Door Restrictions** checked. This allows them to go through in either direction.

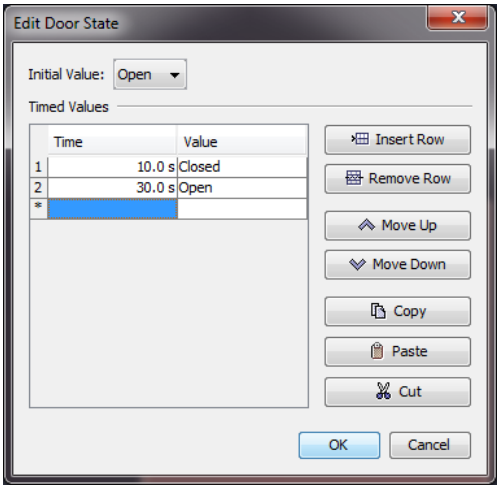
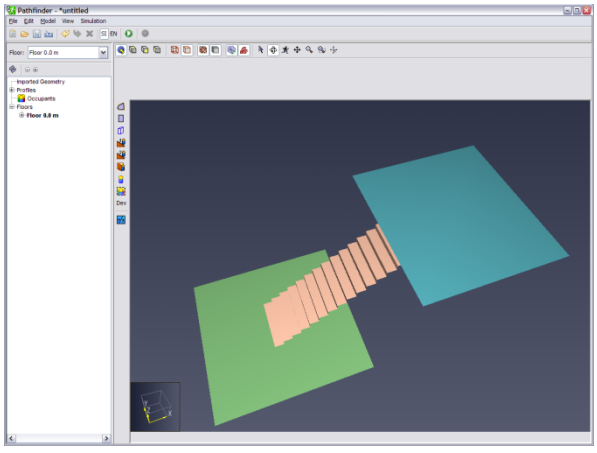


Figure 3-22: Scripted Events dialog

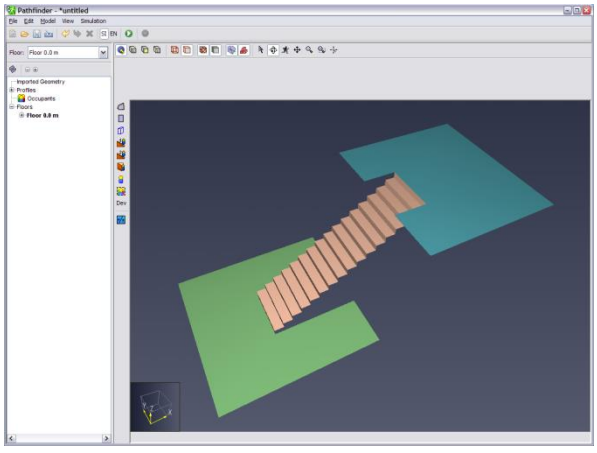
Stairs

Stairs in Pathfinder are represented by one straight-run of steps. They can be created with two tools. One tool allows creation of stairs between two semi-parallel boundaries of rooms, and the other allows creation of stairs that extend from one room boundary until a criterion is met, such as number of steps, height of stairs, etc, or until another room is reached.

One requirement of all stairs for successful simulation is that each end of the stairs must connect to boundary edges of the rooms, meaning that there must be empty space at the top of the stairway and empty space below the bottom. This requirement is shown in Figure 3-23. The size of the gap must be greater than or equal to the radius of the largest occupant to travel on the stairs.



(a) Shows a stair that will not simulate correctly



(b) Shows a stair that will simulate correctly

Figure 3-23: Stair geometry requirements

Stairs between edges

One way to create stairs is to draw them between two pre-existing rooms. Stairs of this type will match the ends of the stairs exactly to the edges that they were drawn between, which means that the tread

rise and run may or may not match the actual slope of the stairs. In Pathfinder, the geometric slope of stairs is unimportant in the simulation, but the specified tread rise and run are.

To create stairs between edges, first ensure that both the connecting rooms are visible. If the rooms are on different floors, at least one room will have to be manually set visible, which can be done through the right-click menu in the Navigation View. Next select the two-point stair tool, . The property panel will display the stair creation properties as shown in Figure 3-24. The stairs can now be created in one of the three following ways:

- **Manual Entry:** Set the desired stair width, and for each edge on which the stairs should be created, enter a point. If the points are valid, a preview stair will be shown in the 2D or 3D view and the **Create** button will enable. Click the **Create** button to add the stairs.
- **Two-click:** Set the desired stair width. Move the cursor over the first edge, and a preview line similar to a thin door will appear previewing the top or bottom of the stairs. Single-click to set the point. Now move the cursor over the second edge. Now a preview of the stairs will appear connecting the first edge to the second. Single-click on the second edge to create the stairs.
- **Two-click with drag:** Click-drag on the first edge to specify both the starting point and width of the stairs. Release the mouse and then single-click on the second edge to place the stairs.

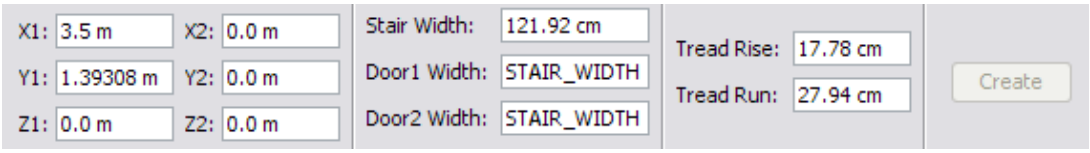


Figure 3-24: Property panel for the two-point stair tool

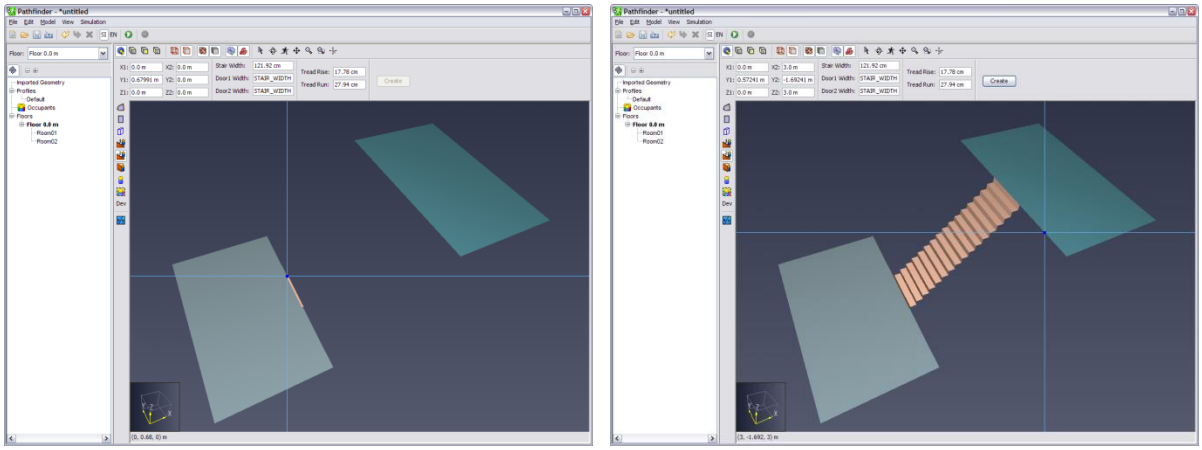


Figure 3-25: Drawing stairs with the two-point stair tool

Stairs extending from one edge

Another way to create stairs is to have them extend from an edge and exactly match the specified tread rise and run. They will stop when they meet a specified criterion or reach another room. The property panel for the one-click stair tool, as shown in Figure 3-26, provides four ways to terminate the stairs:

- **Step count:** The stair will have this many steps.
- **Total rise:** The stair will be this tall in the Z direction.
- **Total run:** The stair will be this length in the XY plane.
- **Total length:** The hypotenuse of the stair will be this length.

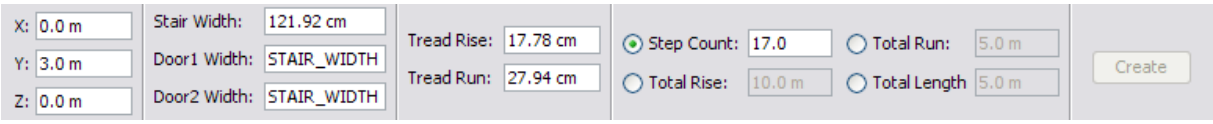


Figure 3-26: Property panel for the one-point stair tool

To create stairs in this manner, select the one-point stair tool, . The property panel will show. If the tread rise is positive, the stairs will extend up from the starting edge, and if it is negative, the stairs will go down. Similarly, if the tread run is positive, the stairs will extend away from the room, and if it is negative, the stairs will extend in toward the room. Another way to change these values is to hold CTRL on the keyboard to make the tread rise negative and hold SHIFT to make the run negative. Now the stairs can be created in one of the three following ways:

- **Manual Entry:** Set the desired stair width, tread rise, tread run, and termination criterion. Specify the starting point on the desired edge. If the location is valid, a preview stair will be shown, and the **Create** button will enable. Click the button to create the stairs.
- **Single-click:** Set the desired stair width, tread rise, tread run, and termination criterion. Move the cursor over the starting point on the room's boundary. A preview stair will be displayed. Single click to place the stairs.
- **Click-drag:** Set the desired tread rise, tread run, and termination criterion. Now click-drag along the room's boundary to specify both the location and the width of the stairs. Release the mouse to create the stairs.

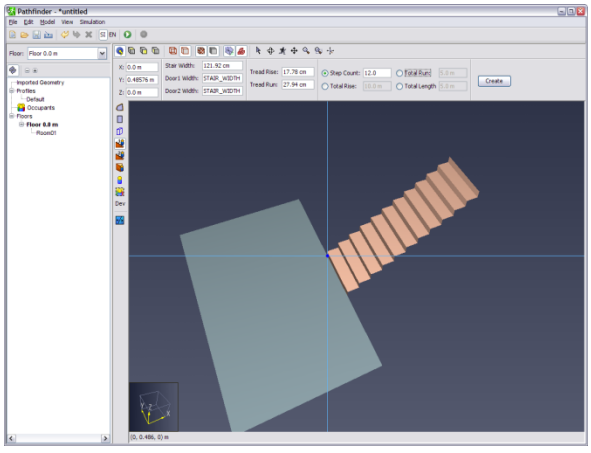


Figure 3-27: Drawing stairs using the one-point stair tool

After creating stairs in this manner, the Z location of the next floor or room will have to match the top of the stairs exactly for the next room to connect properly to the stairs. This can be done by clicking the top of the stairs in the 3D or 2D view when choosing the Z location for either the floor or next room.

Stair properties

Stairs have a number of properties that control their geometry and behavior of occupants that travel on them. When a stair is selected, these properties can be seen in the stair's property panel as shown in Figure 3-28.

Figure 3-28: Stair property panel

- **Riser** and **Tread**: Together, these parameters control the speed at which occupants can travel on the stairs during simulation. While the one-point stair tool uses the tread rise and run to create the initial shape of the stair, these properties can later be changed without affecting the stair shape.
- **Length**: The total length of the stair from the bottom to top edge. This is the same as the hypotenuse formed by the total stair rise and total stair run.
- **Width**: The width of the stair.
- **Top Door** and **Bottom Door**: Clicking these links show the dialog in Figure 3-29. Here, properties of each implicit door can be edited independently of the other door, including **Width**, **Flowrate**, and **State**. For more information on these properties, see Door properties on page 26.
- **One-way**: Indicates whether the stair should only allow occupants to travel in one direction and if so, which direction.
- **Speed Modifier**: A time-variable factor that affects the speed of occupants who travels on the stair. This acts the same as the **Speed Modifier** property for rooms (see Room properties on page 20).
- **Additional Info**: Clicking this link shows additional information about the stair, such as its geometric bounding box, area, and number of occupants.

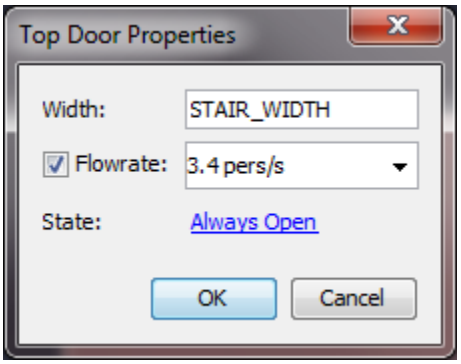


Figure 3-29: Stair door property dialog

Ramps

Ramps are nearly identical to stairs in how they are created and represented. Like stairs, they have two implicit doors at either end and always take the shape of a rectangular piece of geometry. They also have very similar creation tools: the two-point ramp tool, , and the one-point ramp tool, . The key difference between ramps and stairs is that ramps do not affect the speed at which occupants travel.

Escalators

Pathfinder provides some limited support for escalators. They are essentially stairs with slightly modified properties. To create an escalator, perform the following:

1. Create a stair as discussed previously.
2. Select the stair (or several) so its properties are visible in the property panel as shown in Figure 3-28.
3. Set a one-way direction for the stair.
4. Click the **Speed Modifier** drop-down, and choose **Speed Constant** as shown in Figure 3-30.
5. Edit the speed of the escalator in the speed constant field. As with the **Speed Modifier** for stairs, the **Speed Constant** can be time-variable. This would normally be used to turn the escalators on or off throughout the simulation by using the values 1.0 and 0.0 respectively, but any value can be entered.

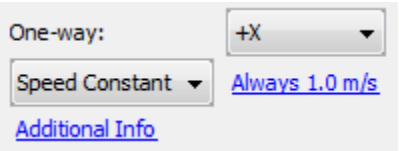


Figure 3-30: Changing a stair into an escalator

In the results view, escalators do not appear differently than stairs.

By default, occupants do not walk on moving escalators. This can be changed by modifying the occupant's profile and selecting **Walk on Escalators** (see Profiles). This causes the escalator's speed constant to be added to the occupant's current speed on the escalator.

NOTE: When escalators are turned off, occupants will use them as stairs, regardless of whether their profile indicates that they want to walk on escalators.

Moving Walkways

Pathfinder also provides limited support for moving walkways. This is similar to creating escalators, but instead of setting a speed constant on an existing stair, the speed constant is set on a flat ramp instead.

Elevators

Pathfinder supports elevators in egress-mode operation, which is based on current thinking described in *Using Elevators In Fires*[1]. The basic operation of elevators in evacuations can be summarized as follows:

- Each elevator has one *discharge* floor. This is where the elevator starts at the beginning of the simulation and where it will take occupants it has picked up.
- Each elevator has at least one *pickup* floor. These are floors where the elevator will pick up occupants that it will take to the *discharge* floor.
- An elevator is *called* on a pickup floor by an occupant when they come within .5 m of the elevator door.
- The elevator uses a priority system to serve *called* floors. By default, floors are served from top to bottom; however, other floors can be given higher priority to simulate fire floors.
- When travelling to a pickup floor, the elevator can change to another pickup floor mid-flight if a higher-priority floor is called that is above the elevator's current location.
- Once an elevator has picked up occupants, it will only travel to the discharge floor before letting the occupants off. It will not travel to any other floor to pick up more occupants.

Creating elevators

Elevators can be made after creating the rest of the model. Perform the following steps to create the elevator (refer to Figure 3-31):

1) Draw a room that defines the shape of the elevator, preferably on the discharge floor.
2) Draw all doors on the boundary of the base room. Occupants will use these doors on every floor to enter and exit the elevator.
3) Right-click the base room, and from the right-click menu, select **Create Elevator…**. This will show the **New Elevator** dialog as shown in Figure 3-32.
4) In the **New Elevator** dialog, enter all parameters for the elevator:
    - **Name** – the name of the elevator
    - **Nominal Load** – the number of people in a full load (estimated). Please read the Nominal Load section below for details.
    - **Elevator Geometry** – the base room that defines the elevator shape. This defaults to the room that was originally selected.
    - **Travel Direction** – a vector defining the direction the elevator can travel. This vector will automatically be normalized. NOTE: the elevator can travel negatively against this vector.
    - **Elevator Bounds** – this defines the bottom-most and top-most floors the elevator can connect to.
    - **Elevator Timing** – this defines a basic timing model used to calculate the travel times for the elevator to travel from the discharge floor to each pickup floor.
    - **Acceleration (Elevator Timing) –** [optional] the acceleration of the elevator.

[1] Richard W. Bukowski and Fang Li, "Using Elevator In Fires," *Consulting – Specifying engineer*, July 21, 2010.

- **Max Velocity (Elevator Timing) –** the maximum velocity that the elevator can accelerate to.
- **Open+Close Time (Elevator Timing) –** the sum of the door opening and closing times. Each value will be taken as half of this.

5) Press OK to create the elevator.

If necessary, Pathfinder will automatically subtract holes in existing geometry to make space for the elevator shaft. It will also delete existing rooms, doors, stairs, and ramps in the elevator shaft. Pathfinder will ask before making any of these changes.

In order for occupants to use elevators, they must explicitly be told to do so through their behaviors, as discussed in Behaviors.

NOTE: The timing values can be re-calculated using a new timing model by selecting the elevator, selecting **Edit** next to **Level Data** in the property panel, and then selecting **Reset…** in the level data dialog.

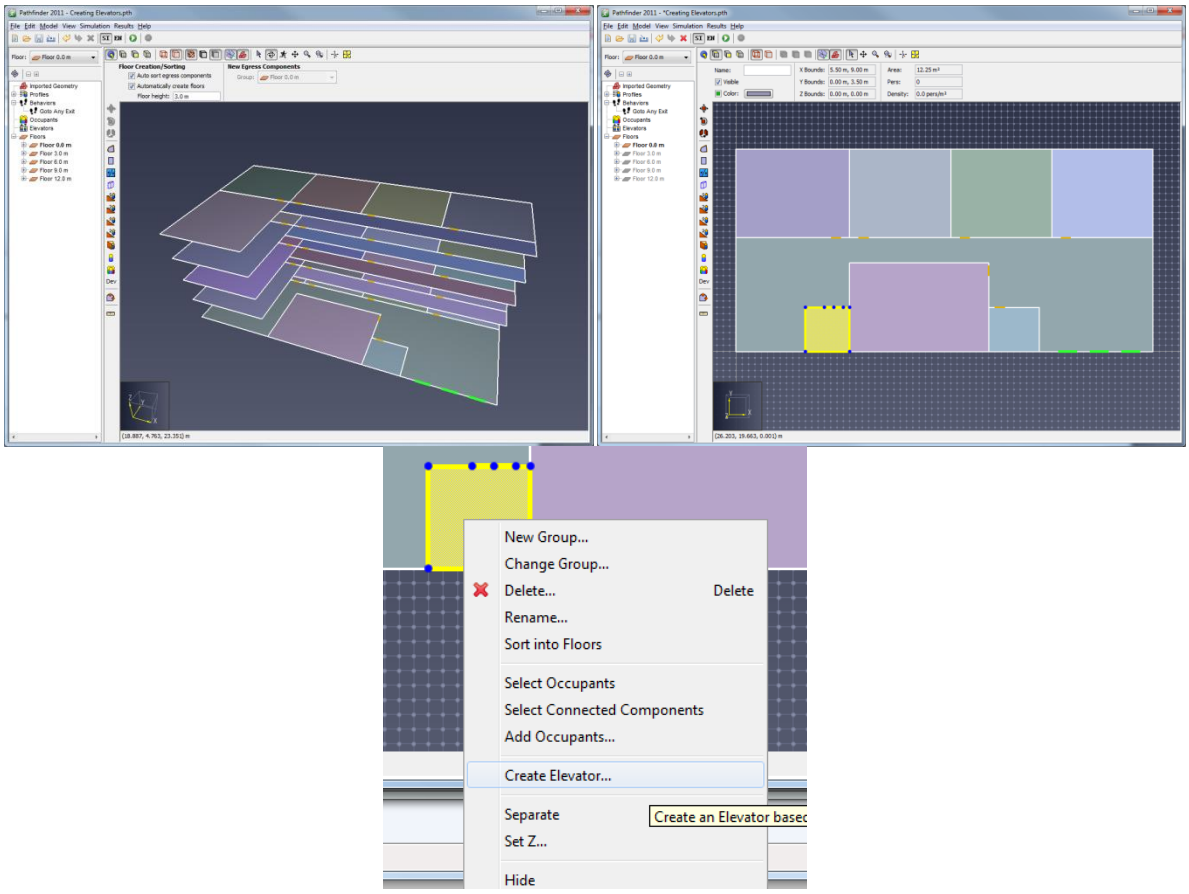


Figure 3-31: Creating an Elevator

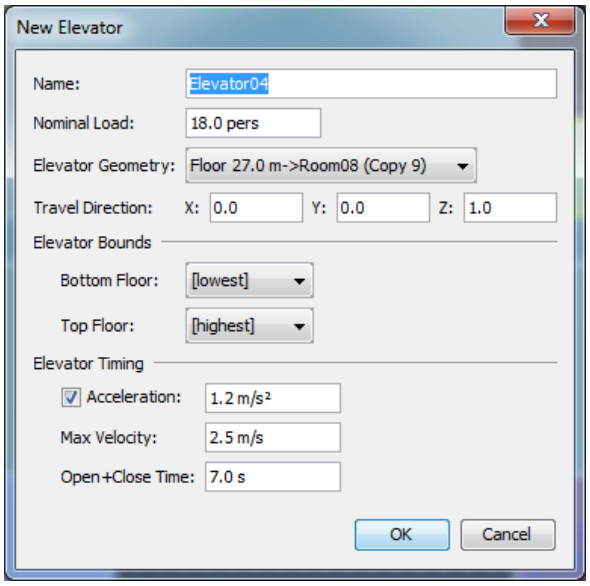


Figure 3-32: New Elevator dialog

Elevator representation

Once an elevator is created, it will appear in the model as a series of "rooms" and doors connected by a transparent elevator shaft as shown in Figure 3-33. There is one room and set of doors for each floor to which the elevator can connect. In the 3D and 2D Views, each room is shaped the same as the base room that created the elevator. In the Navigation View, each room is shown under the elevator rather than the Floors top node. In addition, each set of doors for the room is shown under the room. By default, each of the rooms is named after the floor on which it connects. If the elevator is disconnected completely from a floor as discussed in Connecting/Disconnecting floors, the room is named "<Disconnected Level>".

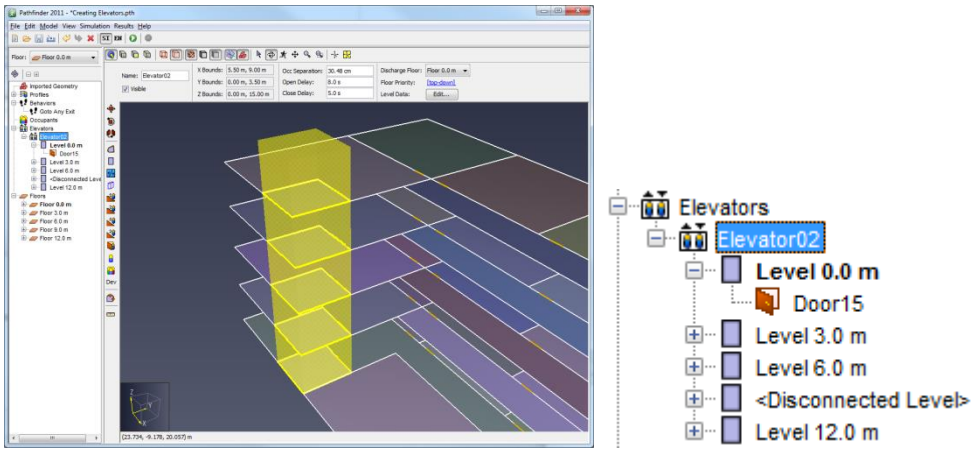


Figure 3-33: Elevator representation

Elevator properties

Once an elevator is created, the elevator's properties can be edited by first selecting it from the navigation view or by ALT-clicking one of its rooms from the 2D or 3D view. Its properties can be edited in the property panel as shown in Figure 3-34.

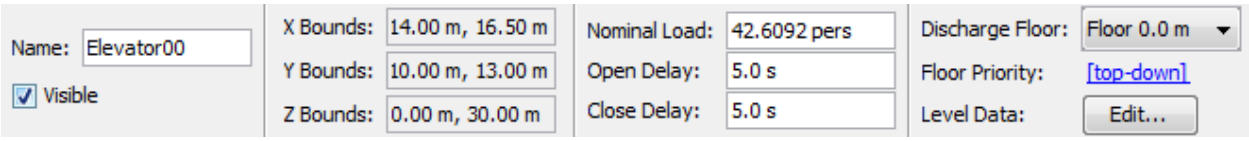

Name: Elevator00
Visible
X Bounds: 14.00 m, 16.50 m
Y Bounds: 10.00 m, 13.00 m
Z Bounds: 0.00 m, 30.00 m
Nominal Load: 42.6092 pers
Open Delay: 5.0 s
Close Delay: 5.0 s
Discharge Floor: Floor 0.0 m
Floor Priority: [top-down]
Level Data: Edit...

Figure 3-34: Elevator property panel

- **Nominal Load** – the number of people in a full elevator load (estimated). Please read the Nominal Load section below for details.
- **Open Delay** – the minimum amount of time an elevator's doors will stay open on a pickup floor.
- **Close Delay** – a delay for closing the elevator doors.
- **Discharge Floor** – the floor at which occupants will discharge during an evacuation.
- **Floor Priority** – the priority of the floors for pickup. By default this is top-down. This can be changed, however by clicking the text, which will show the F**loor Priority** dialog as shown in Figure 3-35. This allows the simulation of a fire floor.
- **Level Data** – click the button to edit timing for each floor. This will open the **Elevator Levels** dialog as shown in Figure 3-36.
    - **Delay** – the time delay from the start of the simulation for when the elevator can start picking up occupants from a floor. This value does not affect the discharge floor.
    - **Open+Close Time** – the total of the door opening and closing time for this floor.
    - **Pickup Time** – the time for the elevator to travel from the discharge to the pickup floor.[1]
    - **Discharge Time** – the time for the elevator to travel from the pickup to the discharge floor.1
    - **Reset** – By selecting the **Reset…** option at the bottom of the dialog, an Elevator Timing dialog will open, allowing automated recalculation of the Level Data parameters. The timing options shown are the same as those shown when creating the elevator.

[1] The pickup and discharge times are calculated from the timing parameters entered in the New Elevator dialog when the elevator is created.

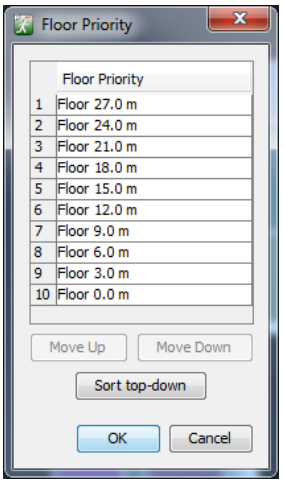


Figure 3-35: Elevator Priority dialog

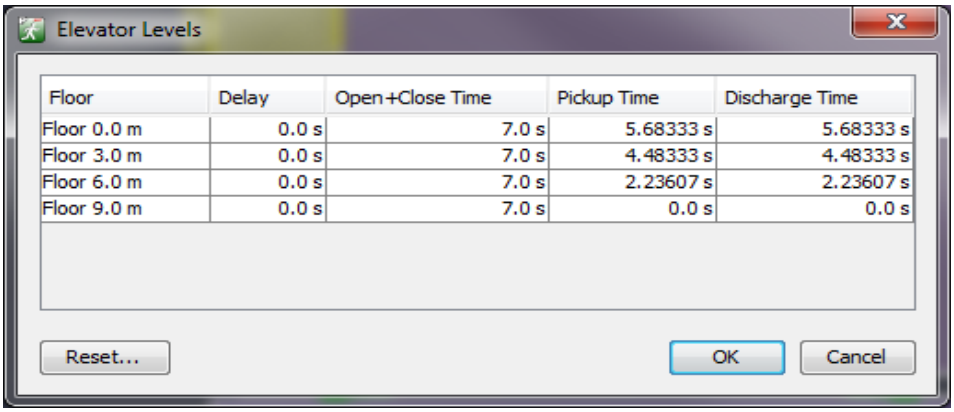

Elevator Levels
Floor | Delay | Open+Close Time | Pickup Time | Discharge Time
Floor 0.0 m | 0.0 s | 7.0 s | 5.68333 s | 5.68333 s
Floor 3.0 m | 0.0 s | 7.0 s | 4.48333 s | 4.48333 s
Floor 6.0 m | 0.0 s | 7.0 s | 2.23607 s | 2.23607 s
Floor 9.0 m | 0.0 s | 7.0 s | 0.0 s | 0.0 s
Reset...
OK
Cancel

Figure 3-36: Elevator Levels dialog

Nominal load

The nominal load is an estimate of the number of people that represent a full elevator load. The default value is based on an estimate of how many occupants of default size (diameter = 45.58 cm) would normally fill the elevator in steering mode. Increasing or decreasing the nominal load will cause occupants' sizes to be scaled up or down while they are on the elevator. The scale factor (default: 1.0) is determined by a correlation to the density produced by the nominal load. This makes it possible to adjust loading while still accounting for differences in individual occupants' size. In steering mode, the geometry of the elevator can lead to reduced loads (e.g. if the elevator is 2.8 persons wide). Please verify that the resulting (post-simulation) elevator loads match elevator manufacturer recommendations.

Connecting/Disconnecting floors

When an elevator is created, by default it is connected to every floor its doors touch along the elevator shaft. Individual elevator doors can be disabled, however, to prevent entering/exiting the elevator through those doors on specific floors. To do so, right-click the elevator door on the desired floor from the Navigation View or 3D/2D View and select **Disable** from the right-click menu. To re-enable it, right-

click the door from the Navigation View and select **Enable**. Alternatively, right-click an elevator level and select **Disable** to disconnect all the elevator's doors on that level, effectively preventing the elevator from picking up occupants on that level.

Call Sets

By default, each elevator is called individually. Elevators can be grouped into call sets, however, so that when one is called, all elevators in the call set respond and travel to the pickup level. To create a call set, create a group in the top Elevators group, and add the desired elevators to the new group. All elevators in the group will be in the same call set.

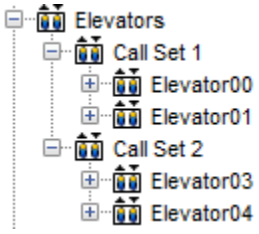


Figure 3-37: Elevator call sets

Exits

In Pathfinder, exits are merely thin doors that exist on the boundary of the model. An exit can only have a room on one of its sides.

Exits are created in almost the same way as thin doors as discussed in the section, Thin doors. The only difference is that the door must lie on an edge of a room, and the edge must not be shared between two rooms.

Exit doors are displayed the same as thin doors except that they are green as shown in Figure 3-38.

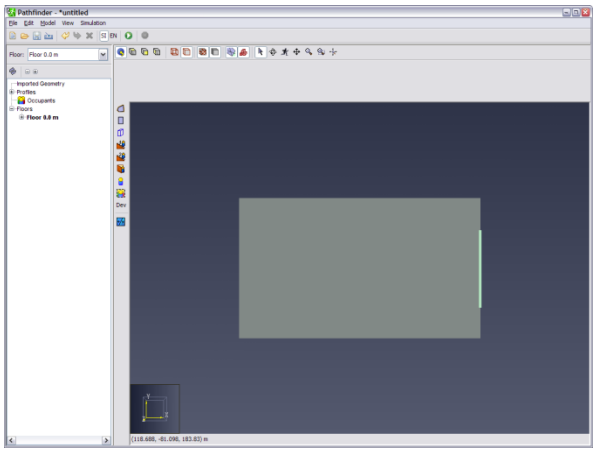


Figure 3-38: An exit door

Importing Files

Pathfinder can import a number of image and geometry files to aid in both drawing and automatically extracting rooms. This includes several image formats, DXF, DWG, FDS, and PyroSim files.

Importing images

Background images can be imported by clicking **Add a Background Image…** on the **Model** menu. When clicked, a dialog will appear prompting for an image file. The following image formats are currently supported:

- GIF
- JPG
- PNG

After a file is selected, a new dialog will appear as shown in Figure 3-39. This dialog allows properties of the image to be specified so that the proper scaling, rotation, and offset can be applied. To specify the scaling for the image, two points, A and B, and a distance between them can be specified. To specify the rotation of the image, an angle can be input that specifies the angle of the vector from A to B with the vector (1,0,0). As shown in the figure, the A->B vector should be 90 degrees from the X axis. The image will automatically be rotated when finished. Finally, an anchor point can be specified on the image that indicates where that point is located in 3D space.

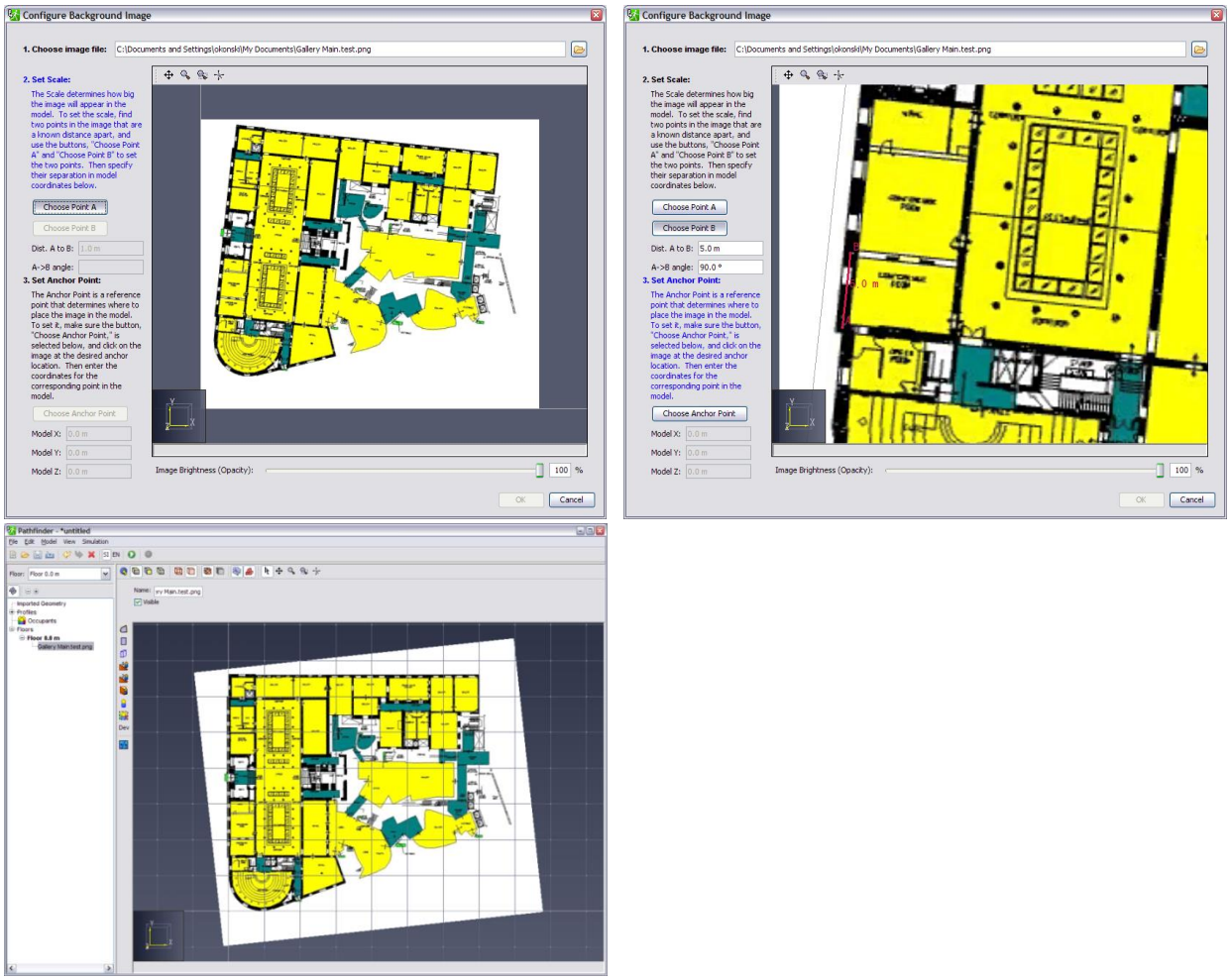

Configure Background Image
1. Choose image file:
C:\Documents and Settings\okonski\My Documents\Gallery Main test.png
2. Set Scale:
The Scale determines how big the image will appear in the model. To set the scale, find two points in the image that are a known distance apart, and use the buttons, "Choose Point A" and "Choose Point B" to set the two points. Then specify their separation in model coordinates below.
Choose Point A
Choose Point B
Dist. A to B:
A->B angle:
3. Set Anchor Point:
The Anchor Point is a reference point that determines where to place the image in the model. To set it, make sure the button, "Choose Anchor Point," is selected below, and click on the image at the desired anchor location. Then enter the coordinates for the corresponding point in the model.
Choose Anchor Point
Model X:
Model Y:
Model Z:
Image Brightness (Opacity):
100
%
OK
Cancel
Configure Background Image
1. Choose image file:
C:\Documents and Settings\okonski\My Documents\Gallery Main test.png
2. Set Scale:
The Scale determines how big the image will appear in the model. To set the scale, find two points in the image that are a known distance apart, and use the buttons, "Choose Point A" and "Choose Point B" to set the two points. Then specify their separation in model coordinates below.
Choose Point A
Choose Point B
Dist. A to B: 5.0 m
A->B angle: 90.0 °
3. Set Anchor Point:
The Anchor Point is a reference point that determines where to place the image in the model. To set it, make sure the button, "Choose Anchor Point," is selected below, and click on the image at the desired anchor location. Then enter the coordinates for the corresponding point in the model.
Choose Anchor Point
Model X:
Model Y:
Model Z:
Image Brightness (Opacity):
100
%
OK
Cancel
Pathfinder - *untitled
File Edit Model View Simulation
Imported Geometry
Profiles
Occupants
Floors
Floor 0.0 m
Gallery Main test.png

Figure 3-39: Importing a background image

The imported image is added to the “Imported Geometry->Background Images” group in the Navigation View. The image can be edited and deleted from there. In addition, any number of images can be added to any floor.

Importing CAD files

Pathfinder can also fully import files saved in AutoCAD’s Drawing Exchange Format (DXF) as well drawing (DWG) files.

To import a DXF or DWG file, under the **File** menu, select **Import…** and select the desired file. After selecting a file, a step-by-step dialog will open as shown in Figure 3-40.

1. **Units**: The first prompt asks the user to select the base unit in which the CAD file was created. If the drawing was saved in a more recent file format, the prompt will default to the stored unit type. The dialog box shows the model’s width, depth, and height based on the selected unit as a guide for selecting the unit. Selecting **Next** brings up the options prompt.

2. **Options:** The options prompt allows the user to specify more options for import. Before this, Pathfinder will attempt to discern if the CAD file contains a 2D Floorplan or a 3D Model and will select default values for the options based on the detected type.
    - **Lines** – check to import lines in the file (default=checked only for floorplans)
    - **Faces** – check to import faces in the file (default=checked only for 3D models)
    - **Move geometry to Z** – if checked, all imported geometry will be offset so that the minimum Z lies in the specified Z plane (default=checked only for 2d floorplans).
    - **Flatten so geometry lies in one plane** – if checked, all geometry will be scaled in the Z dimension by a very small scale (1e-9). This is useful for floorplans that have entities drawn in several planes. This option will flatten them all into one plane. (default=checked only for 2d floorplans).
    - **Add a blank rectangle to obscure lower floors** – if checked, a solid rectangle of the specified color will be added to the model. This is useful to obscure geometry located on lower floors. The imported rectangle will be excluded from floor extraction by default. (default=checked only for 2d floorplans).

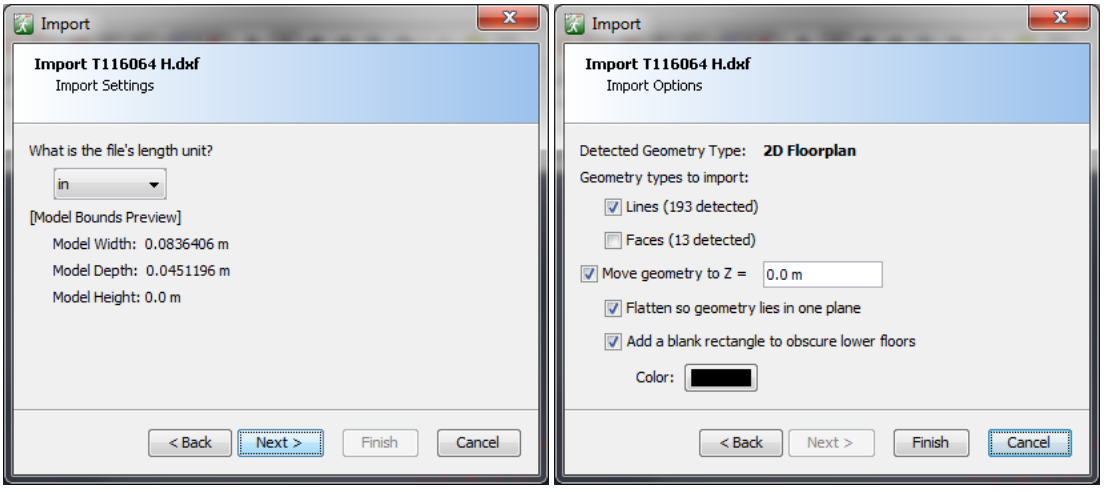


Figure 3-40: Dxf unit dialog box

Choose **Finish** to import the file. All imported elements will be added to the "Imported Geometry" node in the Navigation View. The grouping structure will include the model level, the layer level, and all entities distributed within the layer. If both lines and faces were included in import and an entity contained both lines and faces in the CAD file, the entity is split into two in Pathfinder - one with the lines, and one with the faces.

NOTE: In versions prior to Pathfinder 2012.1, DXF import allowed a background image to be created that could be passed through to the 3D results viewer. While this option is no longer available, both imported line and face data is now sent directly to the 3D results viewer instead, eliminating the need for a background image.

Importing Revit Files

While Pathfinder cannot directly import Autodesk Revit files (RVT), there are two ways to export the data from Revit into a DWG that Pathfinder can read.

- **Revit to DWG (direct)**: The first method is to export a DWG directly from Revit, which can then be imported into Pathfinder. While simple to perform and only requires Revit, this method loses all information about materials, including textures, due to Revit's limited DWG support. To do this in Revit Architecture 2014, perform the following:
    1. Open the desired RVT file within Revit Architecture.
    2. Click the Revit icon at the top left .
    3. Select **Export->CAD Formats->DWG** files.
    4. In the **DWG Export** dialog, for **Export**, select `<In session view/sheet set>`.
    5. For **Show in list**, select `Views in the Model`.
    6. Click the **Check None** button, and then in the view table, select the check box for `3D View: {3D}` (Other views may be chosen, but the DWG will only contain entities visible in the selected views).
    7. Click the **Next…** button and choose a file name for the DWG file.
    8. Click **OK** to create the DWG.
    9. Import the DWG into Pathfinder.
- **Revit to FBX to AutoCAD to DWG:** This method requires both Revit and AutoCAD and does not perform a perfect conversion, but it retains some information about materials and texture coordinates. The steps described here use Revit Architecture 2014 and AutoCAD 2014.
    1. Open the desired RVT file within Revit Architecture.
    2. Click the Revit icon at the top left .
    3. Select **Export->FBX**.
    4. Specify the desired filename and click **Save**.
    5. Open AutoCAD.
    6. On the **Insert** tab in the ribbon, select **Import**.
    7. Select the FBX file created by Revit.
    8. The **FBX Import Options** dialog will appear. The following are recommended settings for the FBX import:
        - **Import section**: Make sure **Objects** and **Materials** are checked. **Lights** and **Cameras** are unused in Pathfinder.
        - **Assign Objects to Layers:** any option may be selected, but **By Material** is a useful option for Pathfinder.
        - **Unit Conversion:** This section is somewhat misleading. While the **Current Drawing Unit** is correct, the **FBX file unit** tends to be incorrect. No matter what unit is displayed in the greyed-out text for **FBX file units**, the actual unit in the FBX file is always `FOOT`. The appropriate values need to be specified to make the proper unit conversion. For instance, if the current drawing unit is `Millimeters`, you can enter the value, `1` on the left and `304.8` on the right because there is 1 foot per 304.8 millimeters.

- **Block:** Uncheck **Insert file as block**.

9. Click **OK** to finish the import. You may receive a warning about the clip plane of the camera.
10. Save the file as a **DWG**.
11. Import the DWG into Pathfinder.

Importing PyroSim and FDS files

Both PyroSim and FDS files can be imported into Pathfinder. To import one of these file types, select **Import…** under the **File** menu. Then select the appropriate FDS or PSM file. The imported geometry will be added to the **Imported Geometry** group.

If the imported file contains holes, the holes will be automatically subtracted from the solid obstructions and discarded. If the file contains grids, the grids will be intersected with each other as FDS would, and the remaining minimum Z faces of the grids will be imported. If the file contains OPEN vents, the vents will be subtracted from the appropriate grid faces and discarded.

Working with Imported Data

Each type of file that can be imported provides an aid for creating navigation geometry. The different types can be worked with in various ways to create the desired rooms, stairs, and doors. Common to all import types is the ability to set some visual properties, such as color and opacity for each component of the geometry. The imported geometry is sent as-is to 3D Results, resulting in a clean and fast graphical representation of the data.

Working with Images

Working with background images requires the user to draw all rooms, doors, and stairs over the background image. Because the drawn navigation geometry will cover the background image, it may be preferable to make the navigation geometry transparent. This can be accomplished by selecting the drawn navigation components and lowering the opacity in the property panel. Figure 3-41 shows a background image with rooms and doors drawn on top, with a lowered opacity for the drawn rooms.

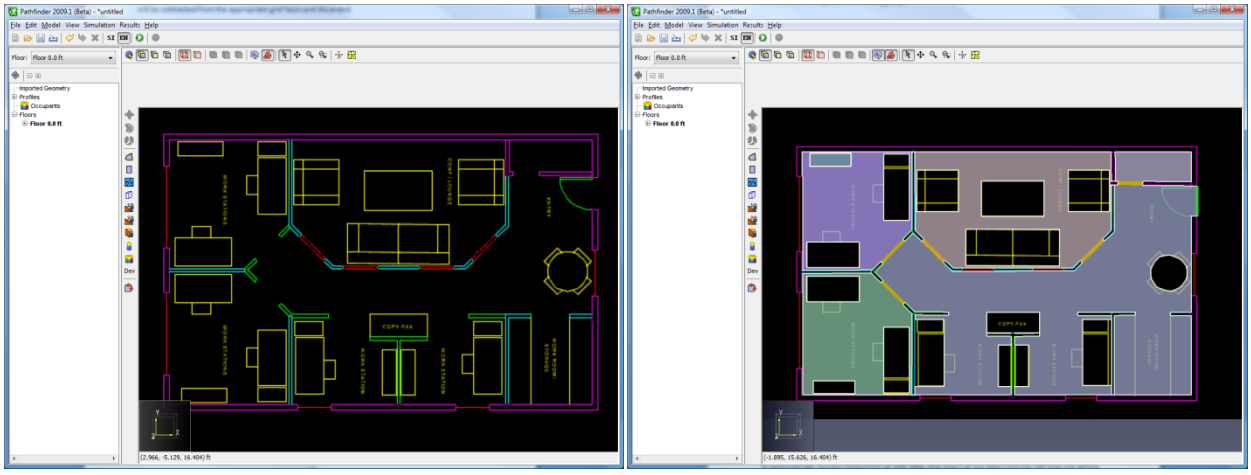


Figure 3-41: Drawing rooms over a background image

To draw rooms on top of a background image, refer to the section, Rooms on page 16.

Working with 3D CAD, PyroSim, and FDS files

Because 3D CAD, PyroSim and FDS files import 3D data, they provide the most helpful information for accurately extracting navigation area. Navigation area can be extracted on a room by room basis using the **Extract floors** tool ( ) from the 3D or 2D views.

The tool properties for this tool are shown in Figure 3-42. The **Max Slope** refers to the maximum grade that a person can walk on. Only imported polygons that have a slope less than this will be included in the result. **Max Head Height** refers to the maximum height of all the agents that will be included in the simulation. This parameter is used to subtract overhead obstructions from the resulting room. The **Gap Tolerance** provides some control in dealing with imperfections in the imported data. If walls are closer than the gap tolerance, Pathfinder will add an extra thin wall in that area, helping to split of rooms so the result does not bleed into undesired areas.

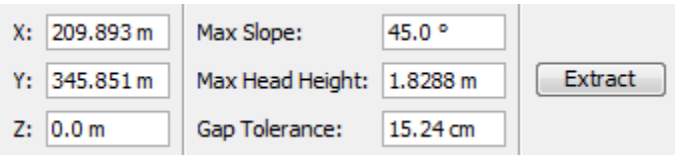


Figure 3-42: Property panel for the floor extraction tool

To extract a room using this tool, first select the tool from the 2D or 3D view. Once the appropriate parameters have been chosen, either enter a location on the floor of the desired room into the property panel or click this point in the 3D or 2D view. If this point does not have any overhead obstructions within **Max Head Height** and the point is on a polygon with a slope less than **Max Slope**, Pathfinder will march out from this point on the imported geometry's polygons until it finds the boundaries of the room. It will also subtract overhead obstructions within **Max Head Height** from the resulting room. An example of an extracted room is shown in Figure 3-43.

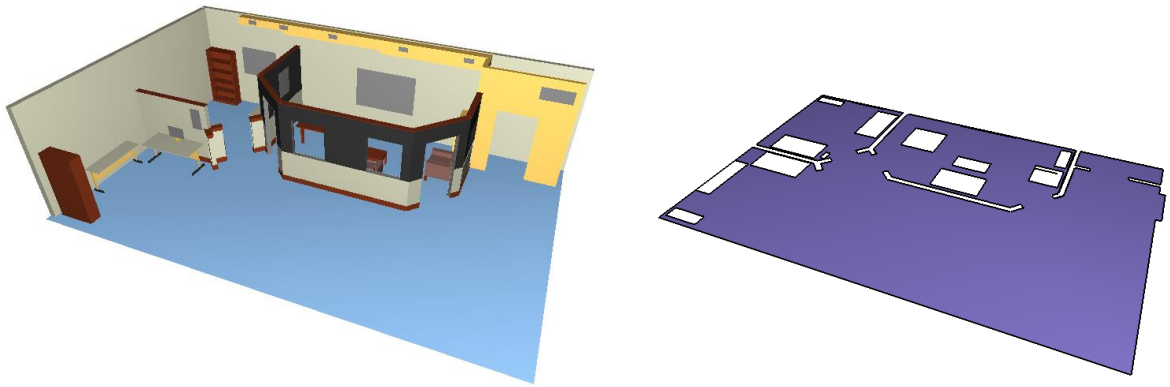


Figure 3-43: A room extracted from a PyroSim file

When using the 3D floor extraction tool, Pathfinder will include all imported geometry with faces, even if hidden. If an object must be excluded from floor extraction, before performing extraction select the imported object and in the property panel, deselect **Include in room extraction**.

Working with 2D DXFs

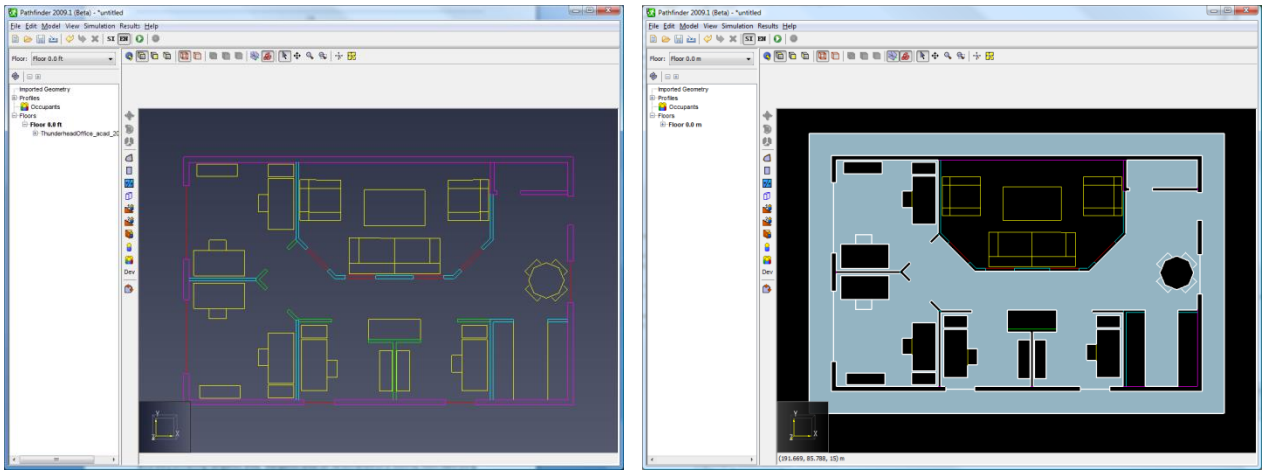
2D DXF data can be worked with in two ways: it can be used as a guide for drawing rooms similarly to background images with the added benefit of being snapped to, or it can have rooms automatically extracted from it similarly to 3D imported geometry.

To sketch the rooms using the built in drawing tools, refer to the section, Rooms on page 16.

Automatic room extraction from 2D DXF data works similarly to extracting rooms from 3D DXF data. As in 3D extraction, the **Extract floors** tool ( ) is used. In addition, the user must click a point in the model with the tool, and one room will be automatically extracted. The main difference from 3D extraction is that the clicked point must not lie on any 3D imported faces marked for floor extraction; instead it must be in empty space (or on the background rectangle imported with a 2d floorplan). The clicked point should also be surrounded by imported 2D lines. These lines will form the boundary of the resulting room. For this reason, any lines that do not contribute to the room's boundary, such as notations, symbols, etc. should be deleted, hidden, or excluded from floor extraction prior to clicking the extraction point. To manually exclude imported geometry from floor extraction, select the geometry and from the property panel, deselect **Include in room extraction**. When determining which imported geometry to extract rooms from, the 2d room extraction tool will automatically exclude hidden objects and those manually excluded.

Once the desired point is clicked, the surrounding 2D lines looking at the model from the **Top View** will be projected along the Z axis onto the active floor's working Z plane. These projected lines will then be used to define the room around the clicked point. If the surrounding lines do not form a closed boundary as shown in Figure 3-44 (a), the resulting room will spill outside of the lines and form a room around the bounding box of all the lines as shown in Figure 3-44 (b). In this case, this outer portion of the room can be separated from the inner portion using the **Thin Wall** tool as discussed in the section, Thin walls on page 18. Once separated, the outer portion can be deleted.

When the extraction tool is finished finding a room, the room will lie in the working Z plane of the active floor.



(a) (b)

Figure 3-44: Room extraction results from an imported 2D Floorplan

Filling in missing pieces

Once rooms have been extracted using the 3D or 2D room extraction tools, the model will still be missing doors and stairs. Doors and stairs must be added manually as discussed in the appropriate sections of this guide.

One feature that may be of particular interest to help this process, however, is the internal door feature of the door tool. This feature automatically finds areas within a room that look like potential doorways and can be used to create a thick door in this area. To use this feature, select the door tool. In the property panel for the tool as shown in Figure 3-18, the **Max Width** refers to the maximum width of the doorway to search for and **Max Depth** refers to the maximum thickness of the doorway. These numbers may need to be larger than for the normal creation of a door to find potential doorways. Once the appropriate parameters have been entered, move the cursor over the desired doorway. A door preview will be displayed. If it doesn't, adjust the search parameters in the property panel and try again. If the door appears correctly, left click the mouse button. The doorway area of the room will be subtracted from the room, and the thick door will be created in its place. The process is shown in Figure 3-45.

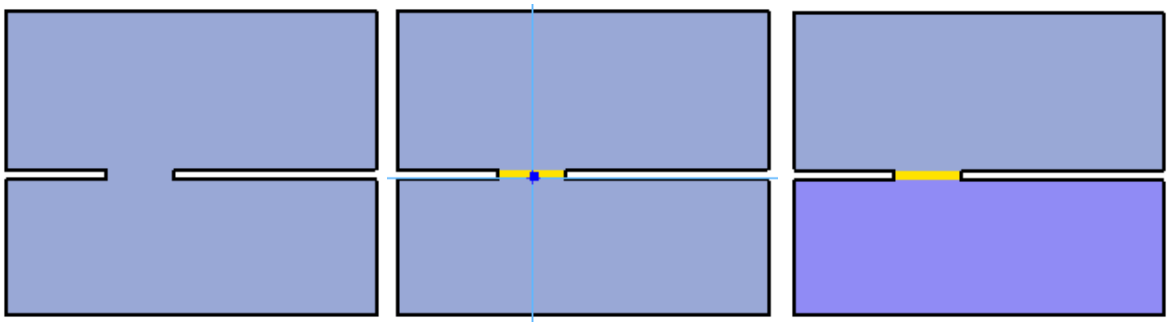


Figure 3-45: Using the door tool to fill a doorway

Flattening and Z Location

Sometimes when importing from 3d CAD models, extracted rooms may not be at the proper height or may have some undesired slope. These issues can be corrected either before or after extracting rooms by using the **Set Z** dialog. To do so, follow these steps:

1. Select the objects that need to be corrected.
2. Right-click the selection and select **Set Z…**.
3. This will open the **Set Z** dialog as shown in Figure 3-46.
4. Set the desired options, and select **OK** to modify the geometry in the selection.

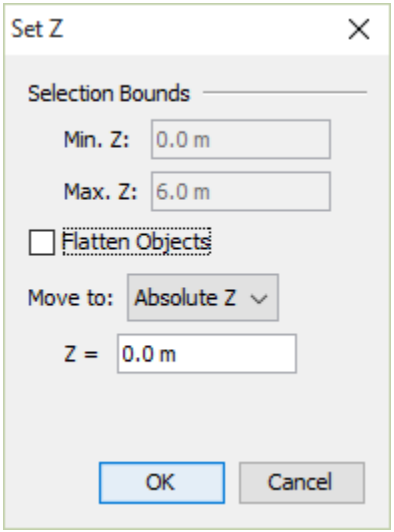


Figure 3-46: Set Z dialog

The dialog shows the following properties and options:

- **Selection Bounds:** Shows the minimum and maximum Z locations of the selected objects.
- **Flatten Objects:** If selected, each object in the selection will be flattened so that all of its geometry lies in the same plane. The location of the plane is determined by the **Move to** option.
- **Move to:** This specifies how the geometry should be moved in the Z direction. This can be one of the following values:
    - **Absolute Z:** All the selected objects will be moved to the same Z plane. If **Flatten Objects** is not selected, each object is moved such that its minimum Z touches the specified plane.
    - **Z Offset:** Each object in the selection is moved by the specified **Z Distance**. If **Flatten Objects** is selected, the object is first flattened to the plane that touches the minimum Z of the object's original geometry, and then moved by the **Z Distance**.
    - **Z Multiple:** Each object in the selection is moved to the nearest multiple of a Z location as specified by **Z Offset** and **Z Multiplier**. The location is determined by the following:

$$z = z_{offset} + z_{multiplier} * n$$

where n is an integer such that the resulting z is the closest to the original object's minimum Z location.

Visual Characteristics

Both 2D and 3D imported geometry have visual characteristics. All geometry types have an object color and opacity. Faces imported from a DWG, PSM, or FDS file may optionally have materials applied to them, which can define more advanced display properties such as texture images as discussed in Materials below. If a face has a material applied, the material's display settings override that of the object color and opacity when the **Show Materials** ( ) option is selected in the **2D View** or the **3D View** (see View Options on page 10).

All display properties can be set through the properties panel when imported geometry is selected, as shown in Figure 3-47.

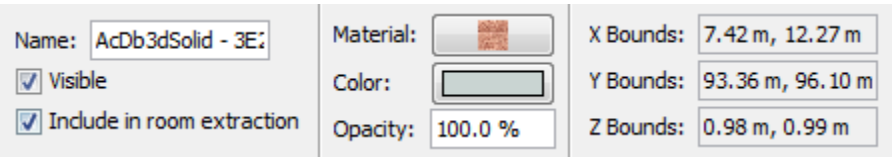


Figure 3-47: Imported Geometry property panel

- **Visible**: Whether the selected object is currently visible.
- **Include in room extraction**: Whether the object should be considered when using the room extraction tool.
- **Material**: The material applied to the selected faces when the **Show Materials** option is turned on. Clicking the material button will open the **Material Dialog** as shown in Figure 3-48 on page 48. In this dialog, the material can either be edited or a new material can be applied to the faces by selecting one from the list on the left and pressing the **OK** button. To remove the reference to the material, select the *<No Material>* option from the material list. NOTE: Because materials can be shared among faces, editing a material applied to one face will also change the visual appearance of all faces referencing that material.
- **Color**: The color of the selected objects when they have no material or the **Show Materials** option is turned off.
- **Opacity**: The opacity of the selected objects when they have no material or the **Show Materials** option is turned off.

Materials

Materials define advanced display properties that can be applied to faces contained in the imported geometry. They are only shown when the **Show Materials** ( ) option is selected in the **2D View** or the **3D View** (see View Options on page 10). Materials can be shared among faces; when a material is edited, all faces referencing that material are updated.

Materials are extracted from import files in different ways, depending on the file type:

- **DWG**: DWG files have a concept of materials. Each material that is referenced by an object in the DWG file will be imported into Pathfinder. Currently, Pathfinder supports the diffuse color, diffuse texture, and opacity setting of the material.
- **FDS and PSM**: Pathfinder materials are constructed from the color and texture settings of the *surfaces* in these file types.
  NOTE: The use of the word "material" in Pathfinder is different than in PyroSim and FDS. In these applications, a material defines the *physical* properties of a substance. In Pathfinder, a material defines the *visual* appearance of a face.

To see the materials that have been imported from the DWG or PSM file, on the **Model** menu, select **Manage Material Database…**. The **Material Dialog** will appear as shown in Figure 3-48.

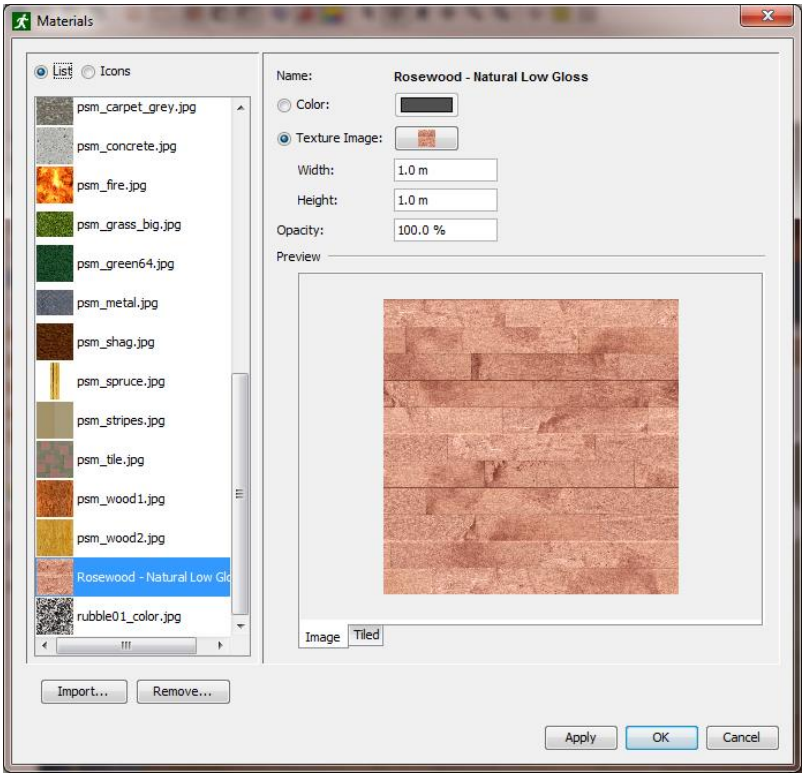


Figure 3-48: Material dialog

Pathfinder provides some default database materials. Most of these materials start with the prefix, *psm_* as in PyroSim. Other materials were either created manually by the user or were imported with the CAD or PyroSim file.

Materials can be added manually by clicking **Import…** under the material list. Currently, a new material must be created from an image on disk that defines the texture of the material. The image is copied into the database directory. Newly created materials are added to the database, and can be used across instances of Pathfinder.

Materials that were imported are only stored in the current Pathfinder file. They are not available when a new model is created, and there is currently no way to import these into the database.

Materials can be deleted by clicking **Remove…** under the material list. If the material exists in the database, all its associated files in the database directory will also be permanently removed.

The following material properties can be edited:

- **Color**: Selecting this option makes the material a solid color.
- **Texture Image**: Selecting this option makes the material show a texture image.
- **Width and Height**: Define the dimensions of the texture image in the model. For instance, if the image depicts a 4x4 array of bricks with no motor between, and each brick is 8”x3”, the **Width** would be 32” and the height would be 12”.
- **Opacity**: Defines the opacity of the material.

Reorganizing and Making Fast Edits

Sometimes the imported data may not be organized in a convenient manner. For instance, it might be desirable to change some feature of all the windows, but the windows in the model may not be in the same group, making it difficult to select all of them at once. In cases such as these, the similar objects may have the same color. If so, right click one of the objects, and choose **Select All by Color**. Alternatively, choose **Select All by Material**. This will find all objects with the same color or material and select them, making it easy to change some shared property or move them into another group for easy selection later.

4. Creating Occupants

In Pathfinder, occupants are defined in two parts: *profiles* and *behaviors*. The *profile* defines fixed characteristics of the occupants, such as maximum speed, radius, avatar, and color. The *behavior* defines a sequence of actions the occupant will take throughout the simulation, such as moving to a refuge area, waiting, and then exiting.

Profiles

Pathfinder uses an *occupant profile* system to manage distributions of parameters across groups of occupants. This system helps you control the occupant speed, size, and visual distributions. To edit occupant profiles, you can use the **Edit Profiles** dialog (Figure 4-1).

To open the **Edit Profiles** dialog: on the **Model** menu, click **Edit Profiles...**

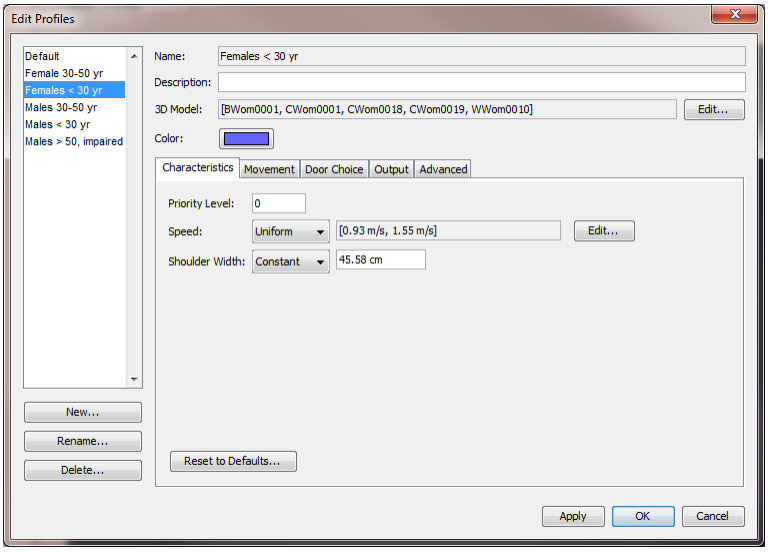


Figure 4-1: The Edit Profiles dialog

The **Description** box provides a place to enter descriptive text. This value is not used outside the **Edit Profiles** dialog.

The **3D Model** input provides a way to use a specific set of 3D human models for an occupant profile. To select 3D models, click **Edit...** on the **3D Model** row. This will open the **3D Models** dialog (see Figure 4-2). When rendering occupants as 3D human models belonging to the current profile, Pathfinder will choose one of the 3D models selected in the **3D Model** dialog. To enable or disable a particular model, click the model's icon.

Figure 4-2: An example of the 3D model dialog.

The **Characteristics** tab provides the following parameters:

- **Priority Level:** the priority of the occupant. Higher values indicate higher priority. This allows occupants of lower priority to move out of the way of those of higher priority. This would be useful when simulating first responders that must be able to move easily through a crowd of occupants. Priority values are completely relative. For instance, if three occupants meet of priorities 4, 6, and 12, they will behave the same as if their priorities were 0, 1, and 2, respectively.
- **Speed:** specifies the maximum speed an occupant may travel in an open room with speed modifier set to 1.0.
- **Shoulder Width:** the diameter of the cylinder representing the occupant. This is used for collision testing and path planning during the simulation. This value will also affect how many occupants can be added to a room without overlapping.

The **Movement** tab provides parameters related to how occupants use their surroundings:

- **Use Stairs:** whether the occupant can use stairs to evacuate. This may be useful to model occupants with physical impairments.
- **Ignore Oneway Door Restrictions:** whether the occupant will ignore the direction specified for one-way doors. If this is unchecked, the occupant will only travel in the direction indicated for the door. If this is checked, the occupant may go either way through any door.
- **Walk on Escalators:** whether the occupant will walk on escalators and moving walkways. If this is checked, the escalator's speed constant will be added to the occupant's desired speed on the stair to determine the occupant's final speed. If unchecked, the occupant will stand still and travel on the stair at the stair's speed constant.

The **Door Choice** tab provides parameters related to how occupants choose doors to exit from in each room. For more information about door choice, see the Pathfinder Technical Reference (Pathfinder Resources).

- **Current Room Travel Time:** a cost factor that affects the cost of travelling to a door in the occupant's current room, ignoring all other occupants. Higher values increase the door's cost in this category.
- **Current Room Queue Time:** a cost factor that affects the cost of waiting in a queue at a door in the occupant's current room. Higher values increase the door's cost in this category.
- **Global Travel Time:** a cost factor that affects the cost of travelling from a door to an exit or the occupant's next goal, ignoring all other occupants. Higher values increase the door's cost in this category.
- **Current Door Preference:** a value used to make occupants stick to their currently chosen doors, preventing excessive door switching. A value of 100% will cause occupants to never switch doors once an initial door is chosen, and a value of 0% will allow occupants to freely change their selected doors.
- **Current Room Distance Penalty:** this value is used to exponentially increase the cost associated with travelling based on how far the occupant has travelled in the current room. This causes the occupant to prefer shorter routes over faster routes the further they have travelled in the current room. Every time the occupant travels this distance in the current room, the travel time cost will have doubled. Setting this value to zero will disable this feature.

The **Output** tab provides the following parameter:

- **Print CSV Data**: When checked, an additional output file will be created for each occupant using this profile. The file contains data for each time step, such as occupant speed, location, etc. For more information, see Occupant History on page 92.
  NOTE: Because this feature creates a file for each occupant with the profile, each containing several pieces of data for each output time step, it may cause the simulation to use significantly more resources, including CPU and disk space. It may be better to only enable this feature for specific occupants. To do this, see Customizing Occupants on page 57.

The **Advanced** tab provides the following parameters:

- **Height:** the height of the cylinder used for inter-occupant collisions. This is useful for limiting collisions that might occur between occupants on different floors when the floors have been modeled close together.
- **Acceleration Time:** a Steering Mode parameter that specifies the amount of time it takes for the occupant to reach maximum speed from rest or to reach rest from maximum speed. The resulting forward acceleration of each occupant is max_speed/accel_time. The occupant uses a separate reverse acceleration of 2*forward_acceleration and a separate lateral acceleration of 1.5*forward_acceleration.

- **Reduction Factor:** a Steering Mode Parameter that specifies how well an occupant may squeeze past others in tight corridors. This factor should be specified as greater than 0 and less than or equal to 1. This factor is directly multiplied by the shoulder width during calculations, so a **Reduction Factor** of 0.5 would lead to the occupant being able to squeeze to one-half his shoulder width.
- **Persist Time:** the amount of time an occupant will maintain an elevated priority when trying to resolve movement conflicts. See the Pathfinder Technical Reference (Pathfinder Resources) for more details.
- **Collision Response Time:** when multiplied by an occupant's current speed, this parameter controls the distance at which an occupant will start recording a cost for colliding with other occupants when steering.
- **Slow Factor:** specifies a fraction of the occupant's speed at which they are considered to be slow. A slow occupant will consider backward directions to separate with others, while a fast moving occupant has a tighter, more focused direction.
- **Boundary Layer:** specifies the distance that occupants try to maintain with walls and other static obstructions.
- **Comfort Distance:** specifies the desired distance one occupant will try to maintain with others in a queue. This may be entered explicitly as a distance, an occupant area, or an occupant density. If entered as an occupant area, the comfort distance is calculated based on sphere packing:

$$c = \frac{2}{\sqrt[4]{12}}\sqrt{a} - d$$

where c is comfort distance, a is the occupant area, and d is the occupant's shoulder width. Comfort distance is calculated similarly when entered as occupant density, except that $a = 1/\rho$ where ρ is occupant density.

NOTE: When comfort distance is entered in terms of occupant area or density, this does not guarantee that occupants will maintain this area or density, but they should be fairly close when occupants are queued.

Each of these parameters (except on/off parameters) can be set using a **constant** value, a **uniform distribution** between two values, or either a **normal** (Gaussian) or **log-normal distribution** using a min, max, standard deviation, and mean.

Each occupant in the Pathfinder model is linked to one profile. Profile parameters can be edited in the profiles dialog at any time and the occupants using that profile will be automatically updated. Occupants' profiles can be set when adding the occupants or by selecting the occupants after being created and editing the **Profile** box in the property panel.

Advanced Speed Properties

In most cases, users only need to enter the maximum speed of an occupant in the **Characteristics** tab of the **Edit Profiles** dialog. The actual speed of the occupant throughout a simulation will vary according to this speed and a set of assumptions from the *Engineering Guide to Human Behavior in Fire* (SFPE, 2003), which takes into account the type of terrain being traversed (stairs, ramps, etc.) and the density of

surrounding occupants. See the Pathfinder Technical Reference (Pathfinder Resources) for more information.

Pathfinder allows for fine-grained control over the speed of the occupant, however, and the SFPE assumptions can be customized or replaced with other assumptions. To do so, follow these steps:

1. In the **Model** menu, click **Edit Profiles…** to open the **Edit Profiles** dialog.
2. Click the **Characteristics** tab.
3. Next to **Speed**, click the drop-down box and select **Advanced** as shown in Figure 4-3.

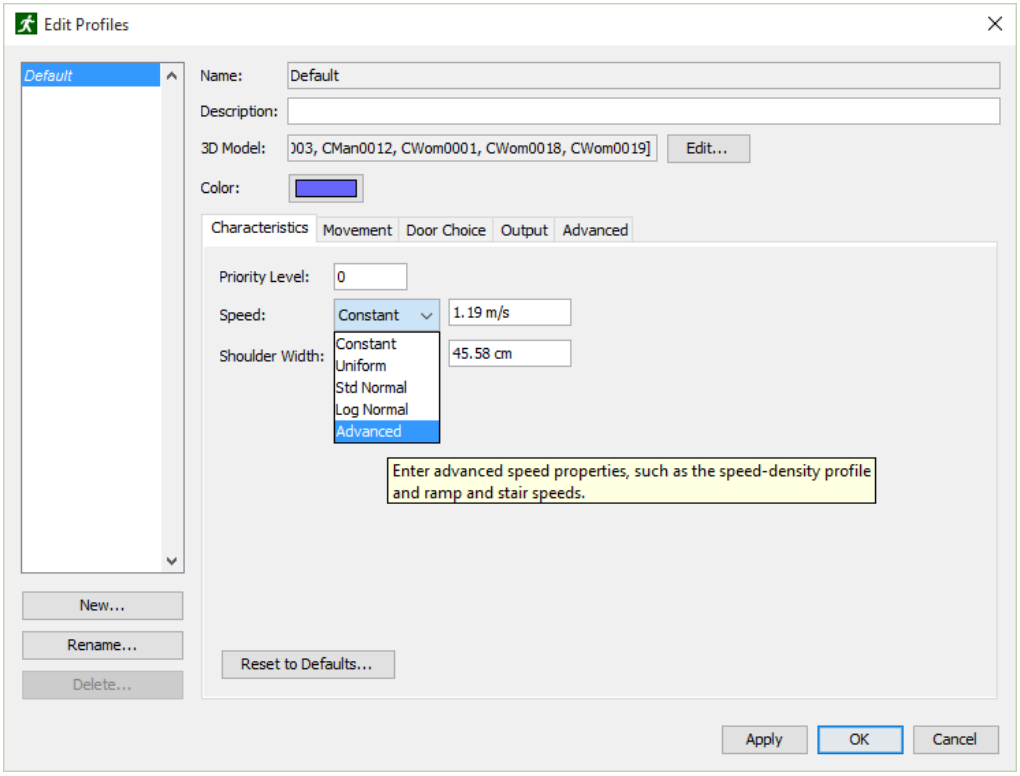


Figure 4-3: Editing advanced speed properties

This will open the **Advanced Speed Properties** dialog as shown in Figure 4-4.

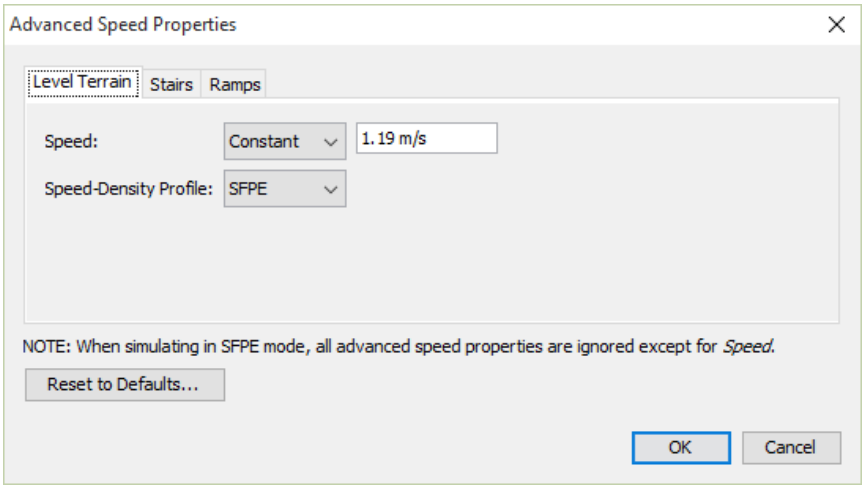


Figure 4-4: Advanced Speed Properties dialog

Each tab in the dialog allows the customization of occupant speed on each terrain type in Pathfinder, including **Level Terrain**, **Stairs**, and **Ramps**.

On the **Level Terrain** tab, the following properties can be modified:

- **Speed:** This defines the occupant's maximum speed. This is the same as the **Speed** property in the **Characterstics** tab of the **Edit Profiles** dialog, only without an advanced option.

- **Speed-Density Profile:** This can be used to set the occupant's speed as a function of the surrounding occupant density, also known as the **fundamental diagram**. One of three options may be chosen for the profile:
    - **SFPE:** The fundamental diagram as specified in the *Engineering Guide to Human Behavior in Fire* (SFPE, 2003) is used to adjust occupant speed.
    - **Constant:** The maximum speed is multiplied by a constant factor to determine the occupant's speed. In most cases, the occupant will either try to move this speed or stop.
    - **From Table:** The speed is entered as a fraction of the occupant's maximum speed as a function of surrounding occupant density. The values are entered in the **Speed-Density Profile dialog** as shown in Figure 4-5. The profile is defined as a piece-wise linear function, where sample points are entered into a table. For densities within the entered range, the speed fraction is interpolated. For densities outside the range, the speed fraction is equal to that entered for the nearest specified density. A preview of the fundamental diagram is shown to the right of the table. The SFPE profile, which is the default profile, may be loaded by clicking **Load SFPE profile…** below the table.
      NOTE: The loaded SFPE profile has a minimum value of 15% of the occupant's maximum speed. This ensures that occupants will not become stuck at high densities.

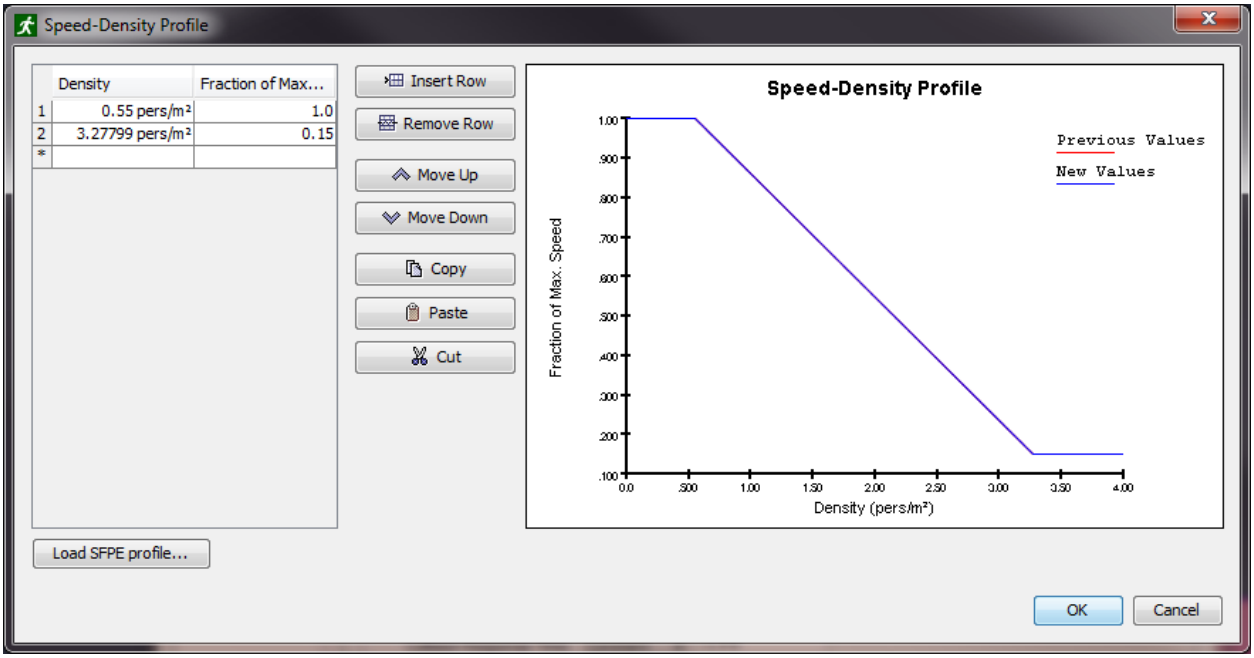


Figure 4-5: Speed-Density Profile Dialog

The following properties may be entered in the **Stairs** tab:

- **Speed Fraction Up**: This defines a speed factor when the occupant travels up stairs. As with the **Speed-Density Profile**, this factor is multiplied by occupant's maximum speed to determine their speed up stairs. This may be entered in one of three ways:
    - **SFPE**: The assumptions from the SFPE *Engineering Guide to Human Behavior in Fire* are used to determine the speed fraction up stairs. This option uses the rise and run of the stair to determine the speed.
    - **Constant**: A constant factor is applied to the maximum speed.
    - **From Table**: The speed factor can be entered as a function of stair slope. Stair slope is defined as step rise/step run. As with the **Speed-Density Profile** when entered from a table, the function is entered as a piece-wise linear function.
- **Speed-Density Up**: This defines the speed-density profile to use when the occupant travels up stairs. This has the same options available as on level terrain, with one additional option, **From Level Terrain**. When **From Level Terrain** is chosen, the occupant uses the same speed-density profile as specified on the **Level Terrain** tab when the occupant travels up stairs.
- **Speed Fraction Down**: This specifies the speed fraction when the occupant travels down stairs. This has the same options available as **Speed Fraction Up**.
- **Speed-Density Down:** This specifies the speed-density profile to use when the occupant travels down stairs. This has the same options available as **Speed-Density Up**.

The **Ramps** tab has nearly identical properties available as on the **Stairs** tab. The only difference is that **Speed Fraction Up** and **Speed Fraction Down** are entered in terms of the ramp's geometric slope rather

than step slope. The geometric slope is determined per triangle in the resulting navigation mesh and depends on the triangle's normal.

As noted in the **Advanced Speed Properties dialog**, when simulating in SFPE mode, only the maximum speed property is used. All others in the **Advanced Speed Properties dialog** are ignored.

Pressing **Reset to Defaults…** in the **Advanced Speed Properties dialog** will reset all advanced speed properties to their default values in Pathfinder.

Seeds

Each occupant has a unique random seed that determines the specific values generated from a profile distribution. Each of these occupant-specific values can be seen by selecting an individual occupant. These specific values will never change unless the distribution is changed in the profile or a new seed is manually generated for the occupant. This ensures that two simulation runs with the same input model will give the same answer. New seeds can be generated for occupants by right-clicking the occupants and selecting **Randomize**.

For an example of the effects of the changing the seed or profile, consider the following scenario:

1. A profile has been created using a uniform distribution of speed on the range 1 m/s to 2 m/s.
2. An occupant is created using this profile.
3. Using the occupant's unique random seed, Pathfinder assigns the occupant a speed of 1.6 m/s based on the occupant's profile.
4. The simulation is run several times, and each time the occupant has a maximum speed of 1.6 m/s.
5. The occupant's profile is changed so that its speed range is .5 m/s to 1 m/s.
6. Pathfinder assigns the occupant a new maximum speed of .6 m/s, which is used for all subsequent simulations.
7. The user randomizes the occupant, and Pathfinder assigns a new maximum speed of .91 m/s to the occupant.

Customizing Occupants

When occupants are selected, their property panel appears as shown in Figure 4-6. Occupants can be given custom profile data once they are added to the model. This can be done by selecting a set of occupants and checking the box next to the parameter to be customized.[1]

When using custom profile data, only constant values can be used for occupant parameters. In addition, once a parameter is customized, any changes to that parameter in the profile will not affect the customized value.

[1] In versions of Pathfinder prior to 2012.1, individual parameters could not be customized. If one parameter was to be customized, all had to be customized.

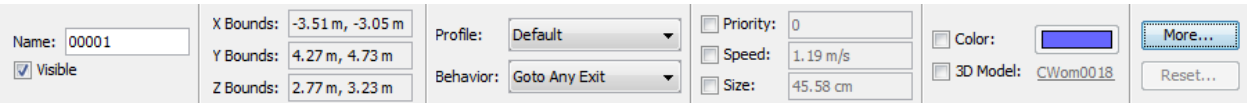


Figure 4-6: Using a custom occupant profile

Occupants with individually customized parameters can easily be found by right-clicking all or a sub-set of occupants and selecting **Select Customized Occupants** from the right-click menu if any exist in the selection.

Behaviors

Behaviors in Pathfinder represent a sequence of actions the occupant will take throughout the simulation. For every behavior, there is an implicit *action* to move the occupant to an exit. This implicit action will always happen last. Additional intermediate actions may also be added that can make the occupant wait or travel to a non-exit destination, such as a room or point. By default, there is one behavior in the model called "Goto Any Exit." This behavior simply makes the occupant move from their starting position to any exit present in the model by the fastest route.

As with profiles, any number of occupants can refer to a single behavior. Any changes to the behavior will be reflected in referring occupants.

Creating a new Behavior

To create a new behavior, right click the **Behaviors** node from the Navigation View, and from the right-click menu, click **Add a Behavior…**, which will open the New Behavior dialog shown in Figure 4-7. In the New Behavior dialog, enter a behavior name, and optionally specify an existing behavior to base the new behavior on. Using this option will copy all the actions from the existing behavior.

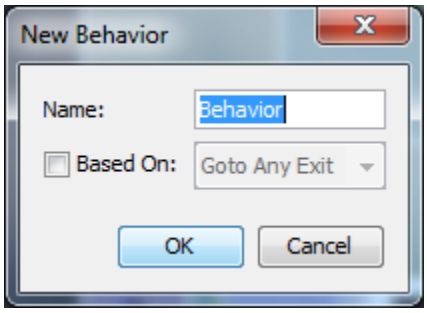


Figure 4-7: New Behavior dialog

With the new behavior selected, the behavior property panel will appear as shown in Figure 4-8.

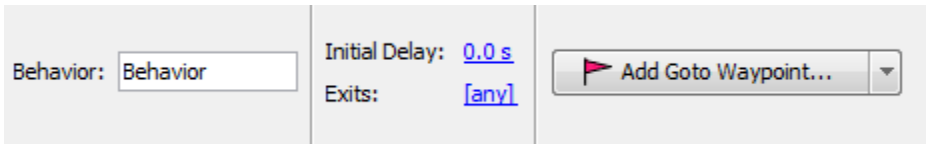


Figure 4-8: Behavior Property Panel

- **Initial Delay** – This specifies an initial delay that makes the occupant wait at their starting position before moving to the next action. If this link is clicked, it will show a dialog where different distribution curves can be entered for the delay, similar to those discussed in Profiles.
- **Exits** – This specifies a set of exit doors the occupant is allowed to exit through during their final implicit exit action. Clicking this link will open the exit chooser, as shown in Figure 4-9.

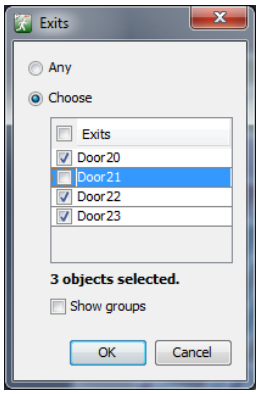


Figure 4-9: Exit Chooser dialog

Adding actions

Additional actions can be added to any behavior, such as going to a room, a waypoint, an elevator, or simply waiting in place. To add an action, select a behavior or existing behavior action. The property panel (Figure 4-8) will show a drop-down button with the description of an action that can be added. To add the currently shown action, simply click the button. To add a different action, click the down-arrow shown to the right of the button and select the desired action from the *behavior actions list*.

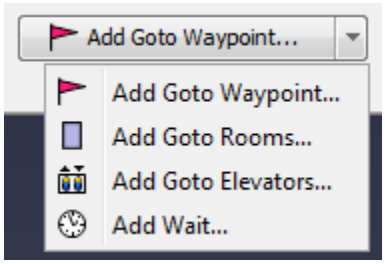


Figure 4-10: Behavior actions list

Once the desired action is clicked, a creation panel will be shown above the 3D/2D View depending on the action. Enter the desired parameters in the creation panel as discussed in the following sections, and then click **Create** to create the action and append it to the behavior. If the behavior itself was selected when adding the action, then the new action will be appended to the end of the list. If, instead, an action was selected when the new action was created, then the new action is inserted directly after that selected action.

Actions always occur in the order shown in the Navigation View. For instance, as shown in Figure 4-11, an occupant using “Behavior1” would first go to any elevator, then go to “Room00”, then wait for 20

seconds, then go to “Room09”, and finally exit. The actions can be reordered at any time (except exiting) by dragging and dropping an action in the list in the Navigation View.

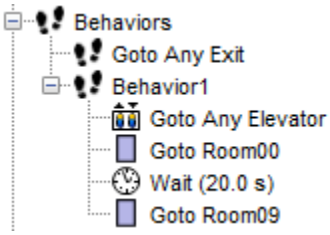


Figure 4-11: Example of action order for a behavior

Goto Waypoint action

A *Goto Waypoint* action specifies that an occupant should go toward a specific point on the navigation mesh. Once they arrive within a certain radius of the point, they will move on to the next action in their behavior.

To add one of these actions, click the **Add Goto Waypoint…** button from the *behavior actions list*. When adding the waypoint, the creation panel will look like Figure 4-12. The **Location** specifies where the occupant must head. This must be a point on the navigation mesh. The **Arrival Radius** defines a circular area around the point. These parameters can be entered manually in the creation panel or be filled in by clicking a point on the navigation mesh in the 3D or 2D View or click-dragging to specify the location+arrival radius. When clicking or click-dragging, the action is created when the mouse button is released.

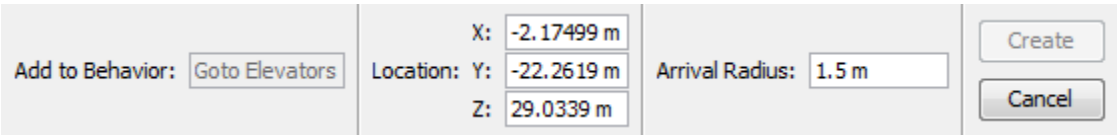


Figure 4-12: Goto Waypoint creation panel

Goto Rooms action

A *Goto Rooms* action specifies that an occupant must select a room out of a set, and go to it. Once they cross a door into the room, they is considered to be in the room and can move on to the next action in their behavior. If multiple rooms are specified for the action, the occupant will go to the one that is fastest for him to reach.

To add a *Goto Rooms* action, click the **Add Goto Rooms…** button from the *behavior actions list*. The creation panel will appear as in Figure 4-13. Either click the **Rooms** link to specify the rooms with a dialog or left-click the desired rooms in the 3D or 2D View. Right-click in the 3D/2D View to finish selecting the rooms and create the action, or click **Create**.

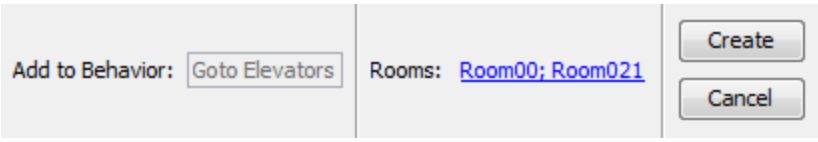


Figure 4-13: Goto Rooms creation panel

Goto Elevators action

A *Goto Elevators* action tells an occupant to use evacuation elevators. When using this action, an occupant will go to a specified elevator, call it, wait for it to arrive, enter it, and then wait for it to reach the discharge floor. Once they reach the discharge floor, they can begin their next action. The *Goto Elevators* action can only be used for occupants who are not on the discharge floor of the desired elevator. If multiple elevators are specified for the action, the occupant will use the one that allows them to reach the discharge floor fastest.

To add a *Goto Elevators* action, click the **Add Goto Elevators…** button from the *behavior actions list*. The creation panel will appear as in Figure 4-14. Either click the **Elevators** link to specify the desired elevators with a dialog or left-click the desired elevators in the 3D or 2D View. Right-click in the 3D/2D View to finish selecting the elevators and create the action, or click **Create**.

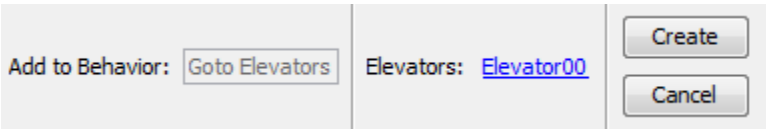


Figure 4-14: Goto Elevators creation panel

Wait action

A *Wait* action tells an occupant to wait in their current location for a certain amount of time. Once that time has elapsed they will begin their next action.

The manner in which they wait will vary depending on their most recent destination. For instance, if their previous destination was a waypoint, they will try to stay close to the center of the waypoint. If the previous destination was a room, they will try to move toward a wall of the room, away from all doors. This allows other occupants to enter the room. If their previous destination is an elevator, they will first move toward the walls as in waiting in a room, and then stand still when the elevator travels. In all cases, a waiting occupant will move out of the way of occupants headed toward a destination unless the destination overlaps with the waiting occupant's most recent destination.

To add a *Wait* action, click **Add Wait** from the *behavior actions list*. The creation panel is shown in Figure 4-15. The **Wait Time** specifies the amount of time they will wait at their current location.

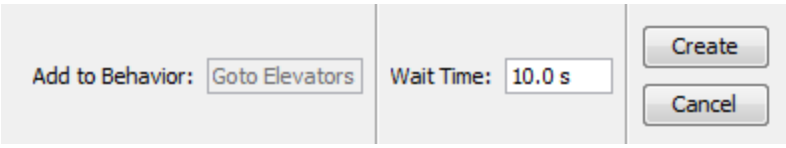


Figure 4-15: Wait creation panel

Adding Occupants

Occupants can be placed individually in the 3D or 2D view, distributed in a rectangular region of a particular room, or distributed through the entire area of a room or multiple rooms.

Individual placement

Individual occupants can be added to the model with the Add Occupant tool, . Occupants can only be placed in pre-existing rooms and stairs and cannot overlap other occupants or room boundaries. Left-click a desired position with the mouse, or enter an x-y-z coordinate and press the Create button from the property panel to place an occupant.

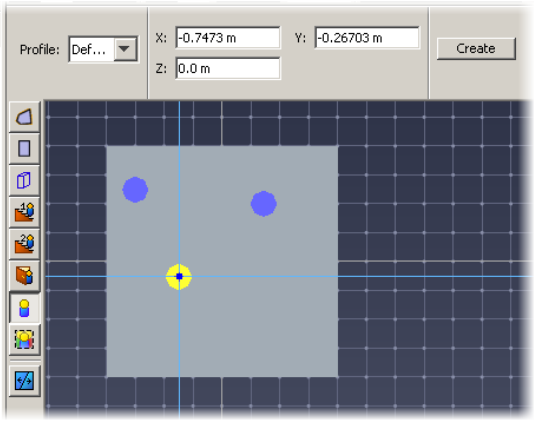


Figure 4-16: Adding occupants individually

Group Placement

Groups of occupants can be added to the model with the Add Occupant Group tool ( ). The occupants are distributed throughout this region using parameters in the property panel as shown in Figure 4-17:

- **Random/Uniform Placement: Random placement** places occupants randomly within the designated area, such that no overlapping of occupants occurs. If the number of desired occupants is too great to accomplish this, a prompt will ask whether or not to continue with overlapping occupants. **Uniform placement** places occupants in an orderly hex pattern, allowing greater occupant densities before overlap occurs. Again, a prompt will ask whether to continue with overlap if the density is too great.
- **Count/Density**: This option specifies whether to place a set number of occupants or place enough occupants to achieve a certain density. Several template densities are provided, and **Custom** can be selected from the densities drop-down menu to enter a new value.

- **Profile**: This option allows a distribution of profiles to be set for the occupants, such as specifying that 25% of the added occupants should be females < 30 years old, 30% children, etc. The label shows the currently set distribution, and if there is more than one profile defined in the model, the value can be clicked to edit the distribution. This will open a dialog as shown in Figure 4-18.
- **Behavior**: Like the Profile option, this option allows the distribution of behaviors to be set.

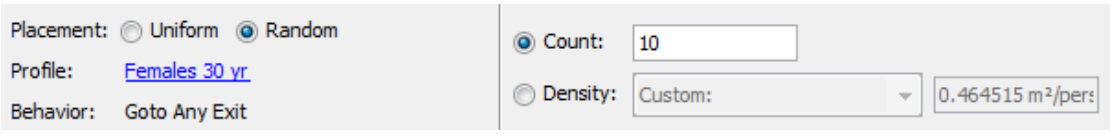


Figure 4-17: Add Rectangular Group of Occupants Property Panel

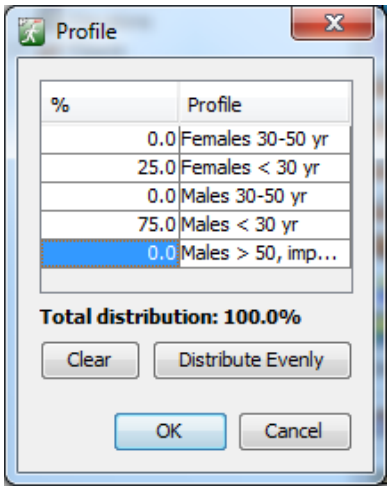


Figure 4-18: Editing the distribution of profiles

Once the properties have been set, click-drag to draw the filling rectangle. Both points of the rectangle must lie in the same room.

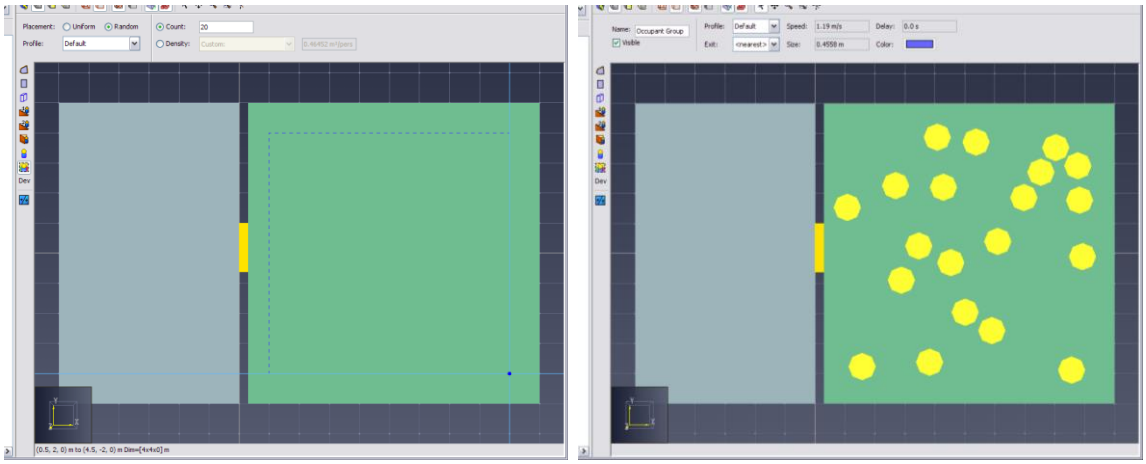


Figure 4-19: Distributing occupants in a rectangular region

When any occupants are selected, the property panel allows the occupants' name, profile, behavior, and color to be edited. The occupants' speed, time delay, and size are also displayed from the occupant's profile, and can be made editable by selecting **Custom** from the profile drop-down menu.

Placement in rooms

In addition to distributing occupants in placement regions, occupants can be distributed throughout entire rooms. To do this, select the desired rooms and choose **Add Occupants…** from the **Model** menu or the right-click menu as shown in Figure 4-20. This will bring up the Add Occupants dialog. For an explanation of the dialog's options, please see the section, Group Placement. Click the **OK** button after selecting the desired options to place occupants and exit the dialog.

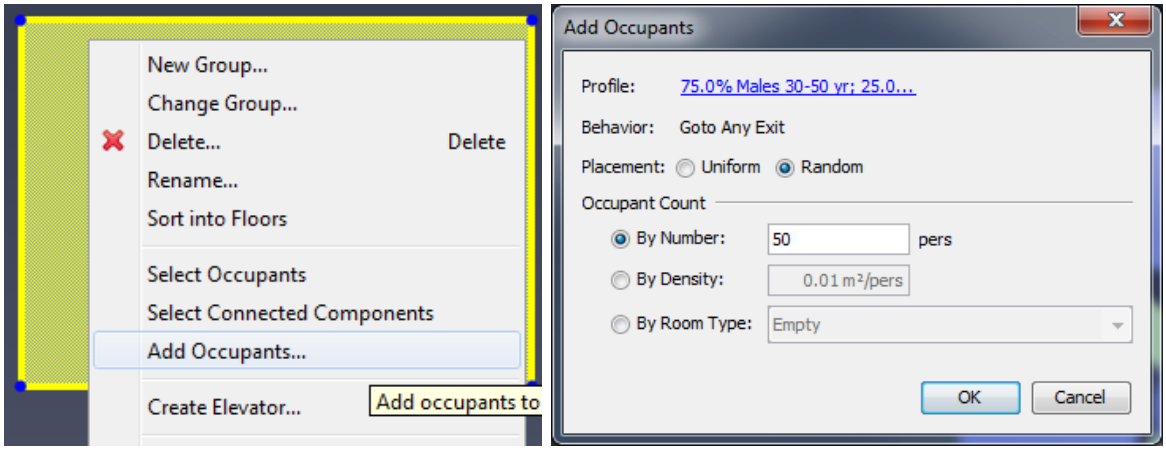


Figure 4-20: Distributing occupants through entire rooms

Redistributing Profiles and Behaviors

Once occupants have been created, the distribution of profiles and behaviors can be reshuffled among them. To do so, select one or more groups of occupants from the Navigation View, right-click them, and from the right-click menu, select **Properties…**, or double-click the occupant group. The Edit Group Distributions dialog will appear as shown in Figure 4-21. If more than one profile exists in the model, the **Profile** link can be clicked to edit the distribution as discussed in Group Placement. If more than one behavior exists in the model, the **Behavior** link can be clicked to edit its distribution.

NOTE: Changing the distribution of profiles or behaviors will not change the number of occupants in the group. It will just change which profile/behavior is assigned to each occupant to match the specified distribution as closely as possible.

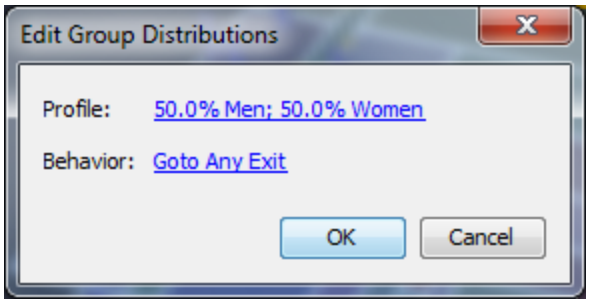


Figure 4-21: Edit Group Distributions dialog

5. Views and Camera Tours

Pathfinder provides the capability to save camera positions (**Views**) that can be recalled later and to create **Camera Tours** that allow flythrough animations of a model. Views and camera tours can be used in both the pre-processor and in the 3D Results.

Views

At any time, the state of the perspective camera can be saved, including its position, orientation, and zoom. The camera information is stored in an object called a **View**. A view can be recalled later in either the pre-processor or 3D Results to view the scene from that perspective.

Views appear in the 3D Results as points and can be hidden either individually or as a whole. To show/hide all views, click the **Show View Objects** button ( ) in the 3D/2D view toolbar.

Creating a View

To create a view, follow these steps:

1. Position the perspective camera using one of the navigation tools as discussed in Navigating the 3D view on page 8.
2. Create a new view using one of these methods:
    - On the **Model** menu, press **New View**.
    - Right-click **Views** in the **Navigation View**, and choose **New View**.

A new view will appear in the Navigation View as shown in Figure 5-1.

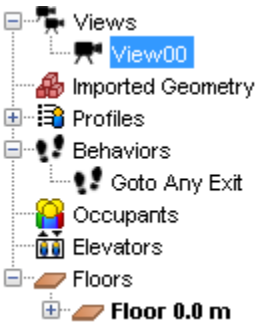


Figure 5-1: Creating a View

Recalling a View

To recall a view in the pre-processor, perform any of the following:

- Double-click the desired view from the Navigation View.
- Right-click a view from either the Navigation View or the 3D/2D View, and press **Show in 3D View**.

This will show the 3D View and the perspective camera will be initialized with the state of the saved view.

Editing a View

If the orientation of a view needs to be changed, perform the following:

1. Position the perspective camera into the new desired position.
2. Right-click the view and press **Update View**.

Alternatively, the graphical representation of the camera can be manipulated to move the location of the view as discussed in Manipulating Objects with Handles on page 75.

Camera Tours

Pathfinder provides the capability to create and view camera tours, which can be used in the following scenarios:

- Create more cinematic views of the results, such as showing a dynamic panning view of the model.
- Simulate security cameras by having the camera pan/tilt back and forth on a timed loop.
- Show several static views of the scene that change periodically.

Camera tour playback can also be recorded in the offline video feature as discussed in High Quality Movies on page 98.

Camera tours show up as green curves in the 3D/2D Views. They can be shown/hidden individually or as a whole by pressing the **Show View Objects** button ( ) in the 3D/2D View toolbar.

Creating Tours from Scratch

Camera tours are built by stringing together several views of the scene and smoothly interpolating between views during playback. Tours may loop and/or repeat and do not have to start at t=0.

There are two ways to create a tour. They can either be created from scratch or they can be created from existing views.

To create a tour from scratch, perform the following:

1. In the **Model** menu, press **New Camera Tour**. This will create an empty camera tour, select it, and show it in the Navigation View. The property panel will appear as in Figure 5-2.
2. Enter the desired properties of the camera tour in the property panel:
    - **Begin Time:** The time at which the tour begins. This delay will only happen once, even if the tour is set to repeat.
    - **Path Generation:** Controls the shape of the camera's path between tour views.
    - **Total Time:** The total time that the tour will take, including the begin time, but excluding repeats.

- **Loop:** If checked, the shape of the path will be a loop, with the first view also being the last.
- **Repeat:** If checked, the tour will repeat indefinitely once it reaches the last view (which may be the first view if **Loop** is checked). When a tour is repeated, the begin time is excluded.
- **Preview:** Previews the tour (see Preview a Camera Tour below).

3. With the new camera tour selected, position the perspective camera in the 3D View to the desired location and orientation. For more information on positioning the camera, see Navigating the 3D view on page 8.
4. In the property panel, press **Add View**. This will add a new tour view to the tour, select it, and show its properties as in Figure 5-3. The new tour view will be initialized to the current state of the perspective camera.
5. Edit the properties of the newly added tour view:
   - **Transition Time:** The amount of time it takes for the perspective camera to transition from the previous view to this view. This value may be zero. In this case, this view will immediately show after the previous view.
   - **Wait Time:** The amount of time for the camera to remain stationary with the current view before it starts transitioning to the next view.
   - **Update View:** Updates this view with the current state of the perspective camera.
6. Repeat steps 3-5 for all the desired views.

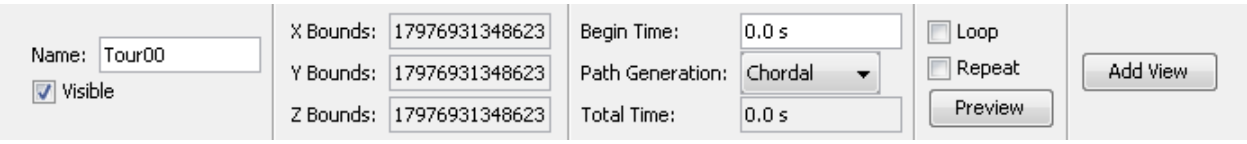


Figure 5-2: Camera Tour property panel

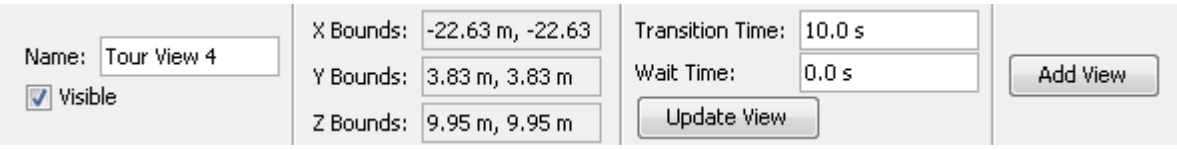


Figure 5-3: Camera Tour View property panel

Creating Tours from Existing Views

To create a tour from existing views, perform the following:

1. Select the desired views in the navigation view or 3D/2D View. The order in which they are selected determines the order in which they will appear in the created tour.
2. Right-click the views and press **New Camera Tour**. This will create a new tour with tour views initialized to the selected views. The existing views will remain in the model, and are independent of the tour views.

Preview a Camera Tour

A camera tour can be previewed in the pre-processor to show all the transitions and the path the camera will take. To preview a tour, perform the following:

1. Select a camera tour either in the 3D/2D view or the navigation view.
2. Press the **Preview** button in the property panel.

Alternatively, double-click one of the camera tours in the navigation view.

The 3D View will show the tour preview playback panel as in Figure 5-4.

Figure 5-4: Tour preview playback panel

This panel allows the preview to be played/paused, skipped backward, rewound, fast forwarded, or skipped ahead, similar to playback in the 3D Results.

To end the preview, either press the **Stop Preview** button in the property panel for the camera tour or perform another action. The view will return to normal.

Editing a Camera Tour

Camera tours can be edited in several ways.

- The order of the tour views can be rearranged by dragging/dropping the views in the navigation view. The order that they appear in the navigation view is the order in which the views will be displayed in the tour.
- A view can be moved to another tour by dragging it to the desired tour in the navigation view.
- Use the manipulate tools to graphically move tour views in the 3D/2D view.
- Delete a tour view to remove it from the tour.
- Insert a new view into a tour at any position:
    1. Select the tour view after which the new view should be inserted.
    2. Position the perspective camera to the desired location.
    3. Press **Insert View** or **Add View** in the property panel (**Add View** only appear if the selected tour view is the last view in the tour. In this case, the new tour view will be added at the end).

Changing Views/Tours without Re-running Simulation

Because views and tours are not simulated objects, they can be changed in the pre-processor after a simulation has been run and the results can be updated without having to re-run the simulation. This might be useful, for instance, if a tour needs to be edited to focus on an unexpected result of the simulation. To do so, follow these steps:

1. Make the desired changes to the view or camera tour in the pre-processor.
2. On the **File** menu, choose **Save Views File…**.
3. Save over the existing input file.
4. Re-load the results in the 3D Results Viewer.

6. Editing and Copying Objects

Most objects can be edited in two ways in Pathfinder. One way is to **transform** the object, including rotating, translating (moving), and mirroring it. Another way is to graphically **manipulate** the objects by dragging **handles**. Objects can also be copied, but currently the only way to do this is through the transform tools as discussed in the following section.

Transforming and Copying Objects

All geometric objects can be transformed and/or copied. All transform/copy options are available through tools in the 3D and 2D views. The following transforms are available and are discussed in the following sections: moving, rotating, and mirroring.

Moving

To move one or more objects, select the objects and click the move tool ( ) from the 2D or 3D view. The property panel for the move tool is shown in Figure 6-1.

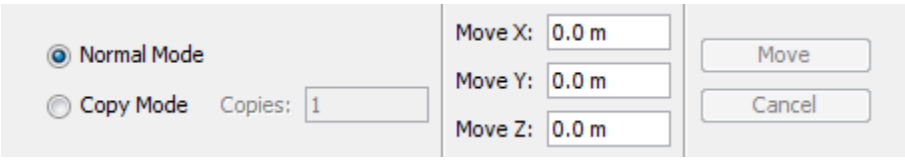


Figure 6-1: Property panel for the translate tool

The object can be moved either manually or graphically:

- **Manually**: Select **Normal Mode** and enter the distance to offset the object in the Move X, Y, and Z boxes. Then click **Move**.
- **Graphically**: This is performed most easily in one of the 2D views. To translate graphically click two points on the model. The vector from the first point to the second defines the movement offset. When moving graphically, objects will only be moved parallel to the camera's view plane. The steps for translating graphically are shown in Figure 6-2.

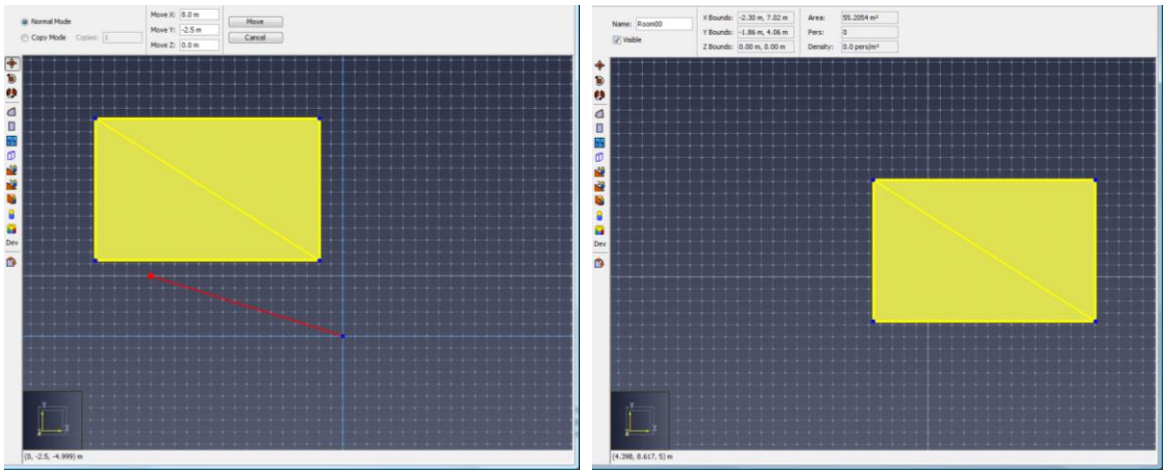


Figure 6-2: Graphically moving an object

Objects can also be copied using the move tool. To do so, select the move tool, select **Copy Mode** from the property panel, and follow the same steps as above for moving an object. Alternatively, hold CTRL on the keyboard while defining the offset. This will create a copy of the object that has been offset by the move distance. Similarly, an array of objects can be made by specifying a value greater than 1 for the **Copies** field in the property panel. The array is created by offsetting each previous copy by the move distance. If, when copying rooms, the resulting copies overlap one another the most recent copies take precedence over earlier ones, meaning that earlier ones will have area subtracted from them. An array is shown in Figure 6-3.

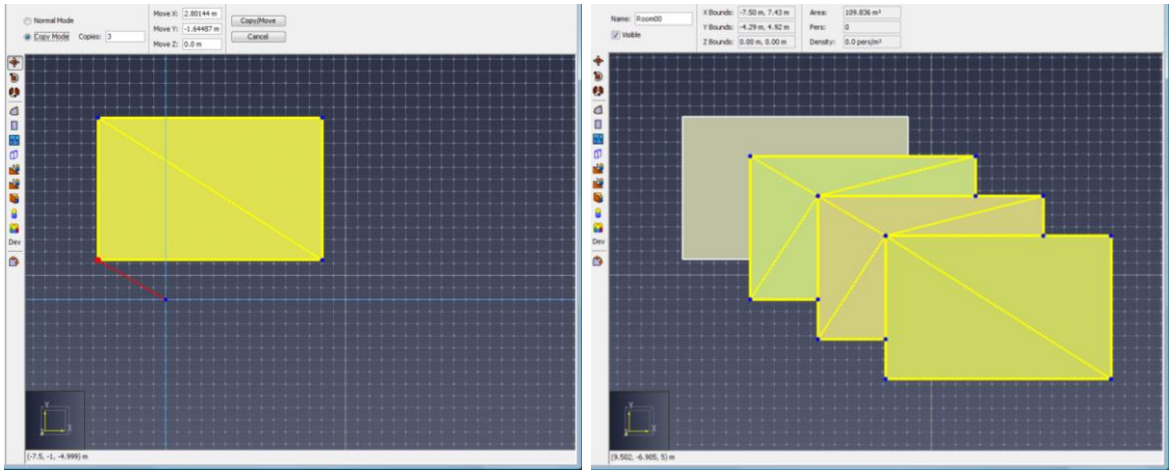


Figure 6-3: Creating an array of objects by using the translate tool

Rotating

To rotate one or more objects, select the objects and click the rotate tool ( ) from the 2D or 3D view. The property panel for the rotate tool is shown in Figure 6-4.

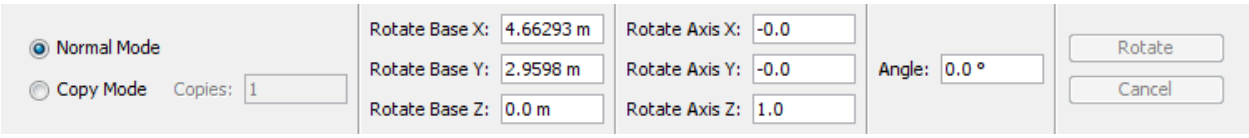


Figure 6-4: Property panel for the rotate tool

The object can be rotated either manually or graphically:

- **Manually**: Select **Normal Mode** and enter the base of rotation, the axis about which to rotate using the right-hand rule, and the angle to rotate. Then click **Rotate**.
- **Graphically**: This is performed most easily in one of the 2D views. The rotate axis is automatically set to a vector normal to the camera. Rotating requires three mouse clicks. The first specifies the base of rotation. The second defines a reference vector extending from the rotation base. The third defines a second vector extending from the rotation base. The rotation angle is the angle between these two vectors. The steps for rotating graphically are shown in Figure 6-5.

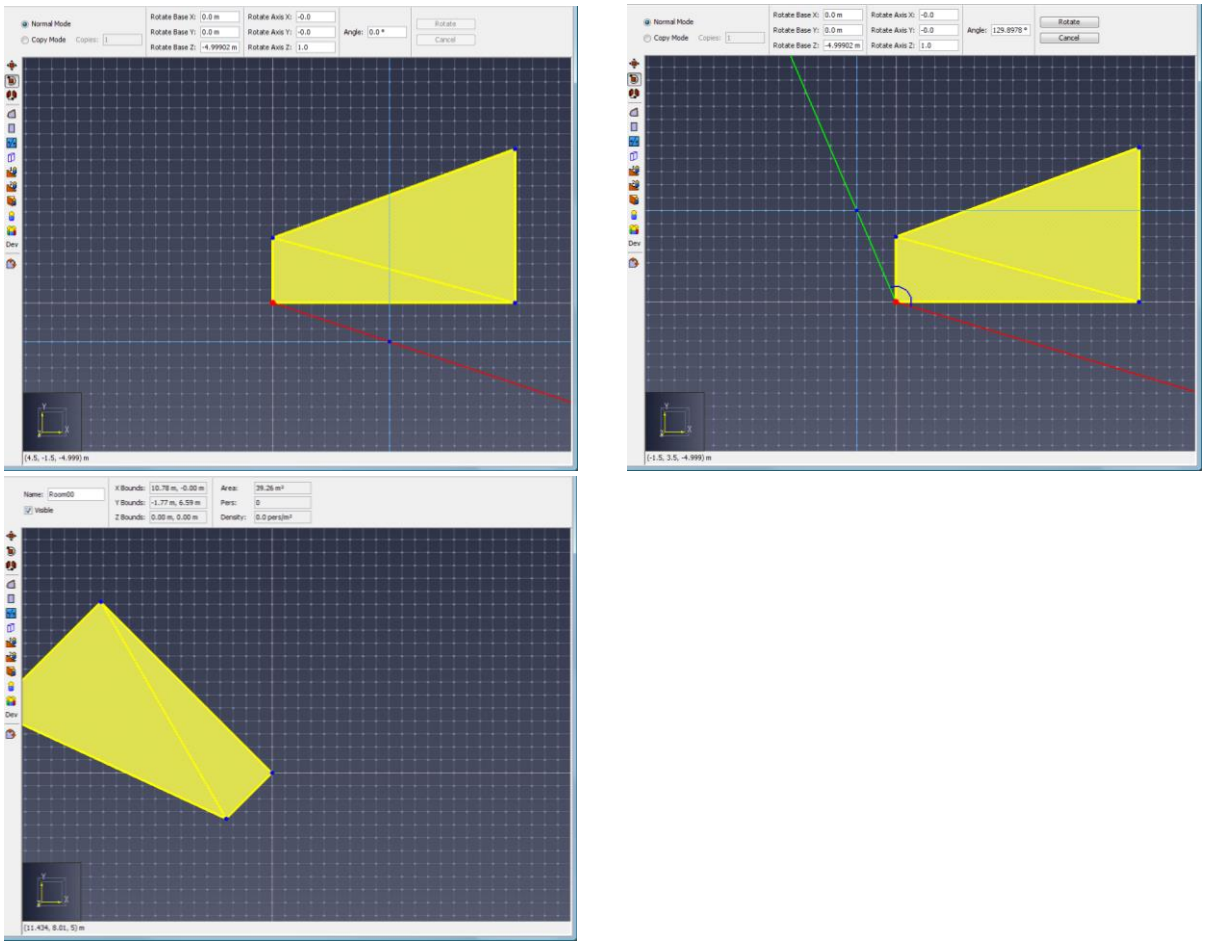


Figure 6-5: Rotating an object

Objects can also be copied using the rotate tool. To do so, select the rotate tool, select **Copy Mode** from the property panel, and follow the same steps as above for rotating an object. Alternatively, hold CTRL on the keyboard while defining the rotate properties. This will create a copy of the object that has been rotated from the original using the rotate parameters. Similarly, an array of objects can be made by specifying a value greater than 1 for the **Copies** field in the property panel. The array is created by rotating each previous copy by the rotate angle. If copying rooms and resulting copies overlap one another the most recent copies take precedence over earlier ones, meaning that earlier ones will have area subtracted from them. An array is shown in Figure 6-6.

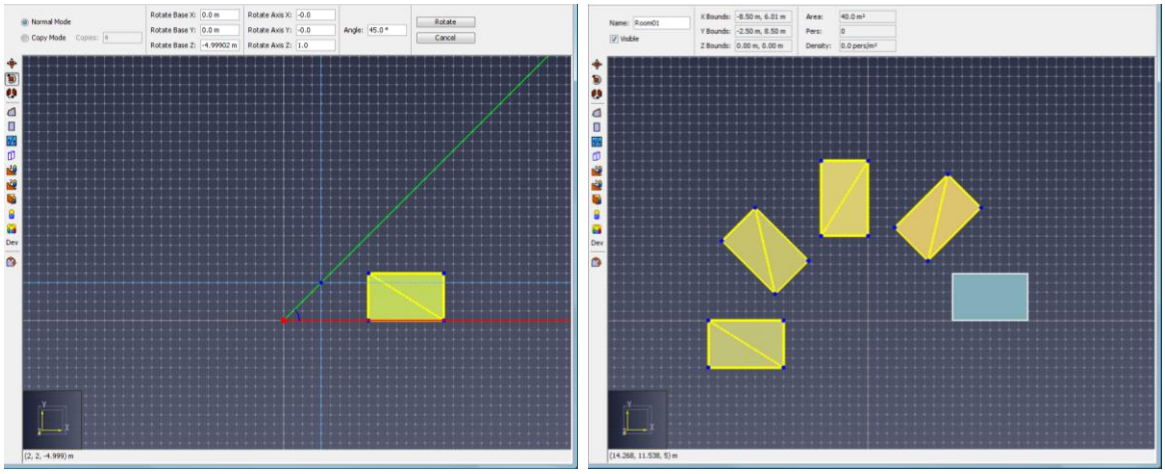


Figure 6-6: Creating an array of objects using the rotate tool

Mirroring

To mirror one or more objects about a plane, select the objects and click the mirror tool ( ) from the 2D or 3D view. The property panel for the mirror tool is shown in Figure 6-7.

Figure 6-7: Property panel for the mirror tool

The object can be mirrored either manually or graphically:

- **Manually**: Select **Normal Mode** and enter the plane about which to mirror. This can be an axis-aligned plane or a custom plane specified by the plane equation, ax + by + cy + d = 0. Next click **Mirror**.
- **Graphically**: This is performed most easily in one of the 2D views. The mirror plane is always perpendicular to the camera's view plane. Defining the plane requires two mouse clicks that define two points in the plane. The steps for mirroring graphically are shown in Figure 6-8.

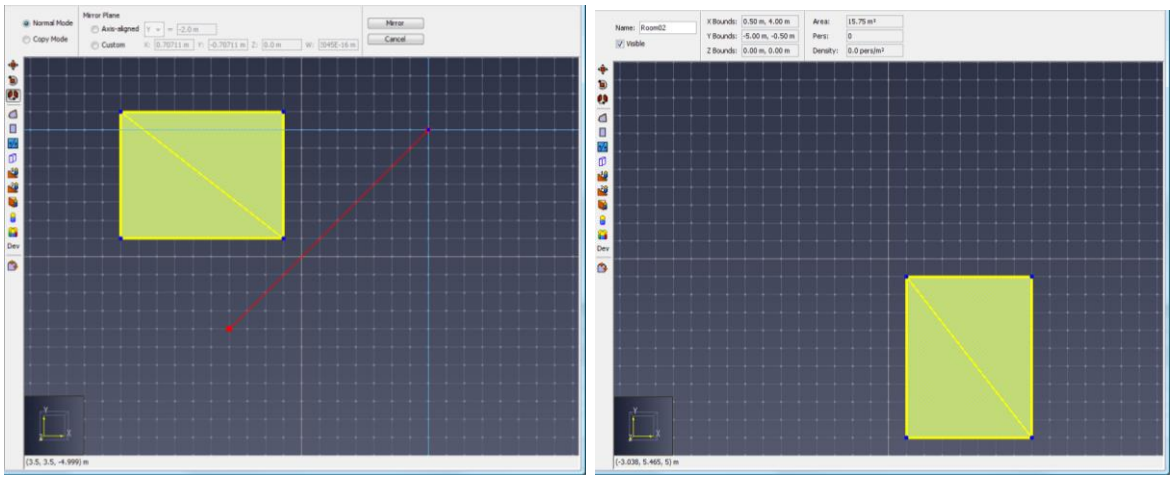


Figure 6-8: Mirroring an object

Objects can also be copied using the mirror tool. To do so, select the mirror tool, select **Copy Mode** from the property panel, and follow the same steps as above for mirroring an object. Alternatively, hold CTRL on the keyboard while defining the mirror plane. This will create a copy of the object that has been mirrored from the original using the mirror plane.

Manipulating Objects with Handles

Some objects, including occupants, rooms, stairs, and doors, can be edited through manipulator **handles**. **Handles** act as points on an object that can either be dragged with the selection tool or edited by the keyboard to edit the attached object. Handles only appear on selected objects and show as blue dots as shown in Figure 6-9.

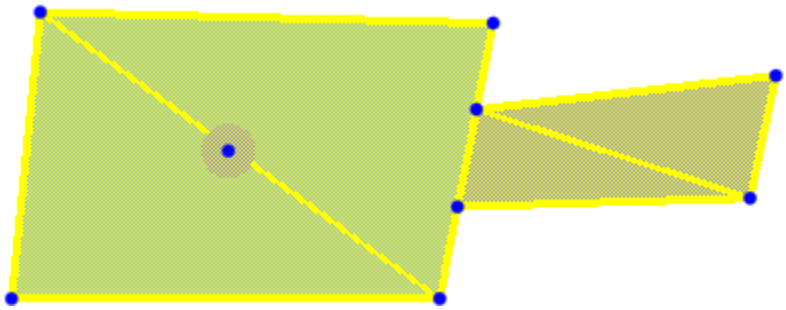


Figure 6-9: Manipulator handles

Selecting and deselecting a handle

To select an object's handle, the object itself must first be selected. Once it is selected, the blue handles should appear. Next select the **Select/Edit** tool ( ). Now an individual handle can be selected by clicking it, which will make the handle property panel appear as shown in Figure 6-10. To deselect the handle, press Escape on the keyboard, click anywhere else in the model, or select another object.

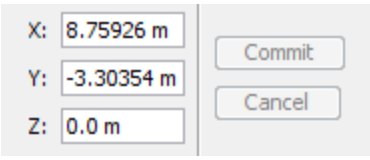


Figure 6-10: Handle property panel

Editing a handle

A handle can be edited in one of two ways: it can be edited with the keyboard to enter precise values or it can be edited graphically.

- **Editing with the keyboard**: to edit with the keyboard, a handle must first be selected. Next enter the desired location in the X, Y, and Z fields in the property panel and select the **Commit** button. The handle will attempt to modify the underlying object using the handle's internal constraints that are described in the appropriate section of this guide for the respective handle's object.
- **Editing graphically**: a handle does not have to be selected before editing graphically. To edit graphically, make sure the **Select/Edit** tool is selected, and press the left mouse button over the desired handle and drag the handle to the desired location. Release the left mouse button, and the object will be edited. A real-time preview of the object being edited will be shown as the mouse is dragged.

Room handles

When rooms are selected, a handle can be found at every vertex on the boundary of the room. The handles move the underlying vertex to reshape the room. The handles can be moved to any location within the plane of the face the vertex is on. If the vertex is shared between two faces in non-parallel planes, the handle can only be moved along the edge to which it is attached.

Thin door handles

When a thin door is selected, three handles will be displayed as shown in Figure 6-11 (a). The handles on the ends of the door allow the door to be moved along the edge to which it is attached. The middle handle allows the door to be made thick by moving the handle to the edge of another room as shown in Figure 6-11 (b).

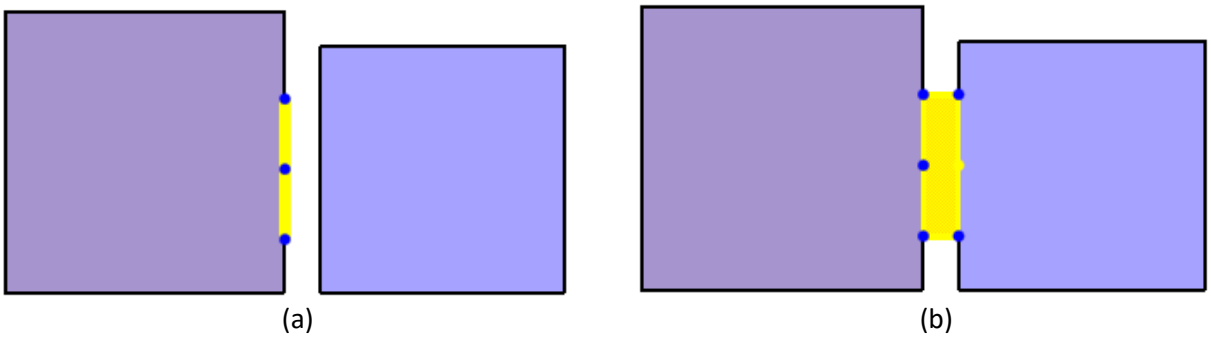


Figure 6-11: Door handles

Thick door handles

When a thick door is selected, six handles will be displayed as shown in Figure 6-11 (b). The four handles on the corners of the door allow the door to be moved along the edge to which each handle is attached. Each middle handle allows the door to be made into a thin door by dragging to the other middle handle. The middle handle can also be useful to reattach the door to a room if the door somehow became detached (such as by a modification to the room).

Stair and Ramp handles

When a stair or ramp is selected, six handles will be displayed as shown in Figure 6-12. The four corner handles each allow the stair/ramp to be moved along the edge to which the handle is attached. The middle handles allow the stair/ramp to be reconnected to another room. The middle handles can also be useful if the geometry of one room attached to a stair/ramp has changed in such a way that the stair/ramp is no longer attached to the room. The middle handle can be used in this case to reconnect to the room.

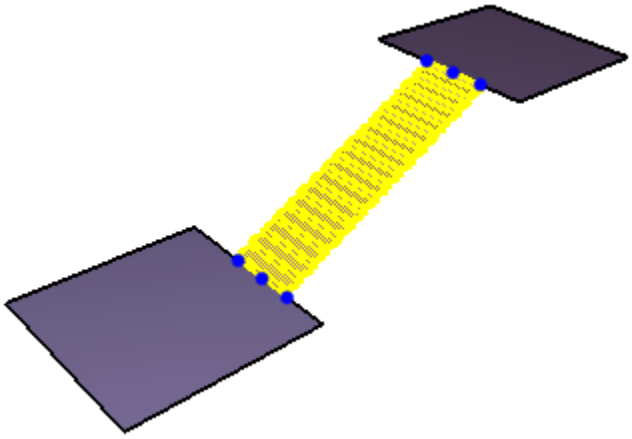


Figure 6-12: Stair/ramp handles

Occupant handles

When an occupant is selected, there is only one handle as shown in Figure 6-13. The sole purpose of this handle is to move the agent to another location. Moving an agent in this manner has a benefit over the translation tool in that the location automatically snaps to an existing room or stair as when adding an agent using the agent dropper tool.

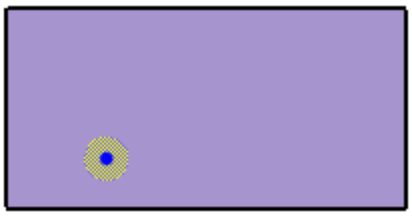


Figure 6-13: Agent handles

Waypoint handles

When a waypoint is selected, five handles are displayed as shown in Figure 6-14. The center handle allows the waypoint to be moved to another location similarly to the occupant handle. The four handles on the perimeter of the circle can be used to change the arrival radius of the waypoint.

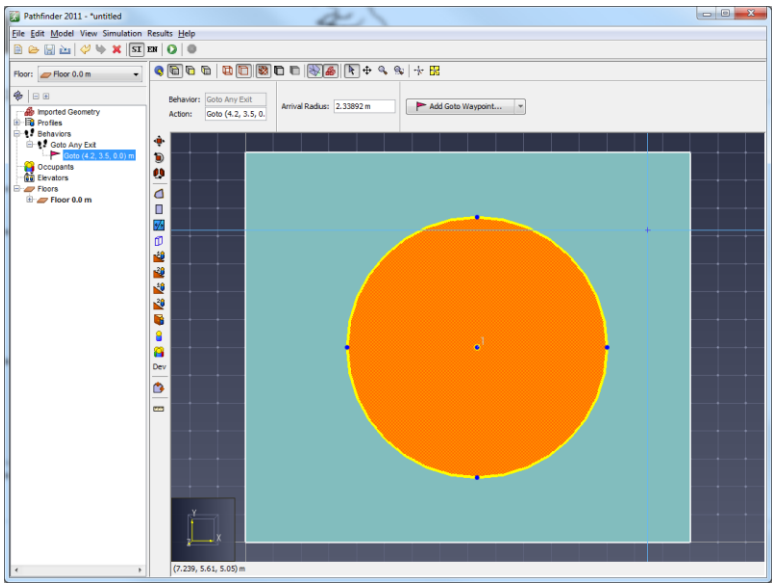


Figure 6-14: Waypoint handles

7. Model Analysis

Pathfinder contains some useful tools to analyze various properties of a model.

Measuring Distances

Distances can be measured by using the measuring tool (). To do so, select the measuring tool from the 3D or 2D view. Left-click a series of points to define a path along which to measure a distance. When done defining points, right-click. The cumulative point-to-point distance is displayed in a dialog box.

When measuring distances in the 3D view, the distance is taken as the actual distance between snapped points. When measuring distances in the 2D view, however, the distance is taken by projecting the points onto a plane parallel to the camera view plane, and then taking the distance.

Checking Connectivity

Sometimes it is desirable to check how various components are connected in the model to debug simulation errors or to ensure model validity. For instance, a model might contain two rooms connected by a hallway as shown in Figure 7-1.

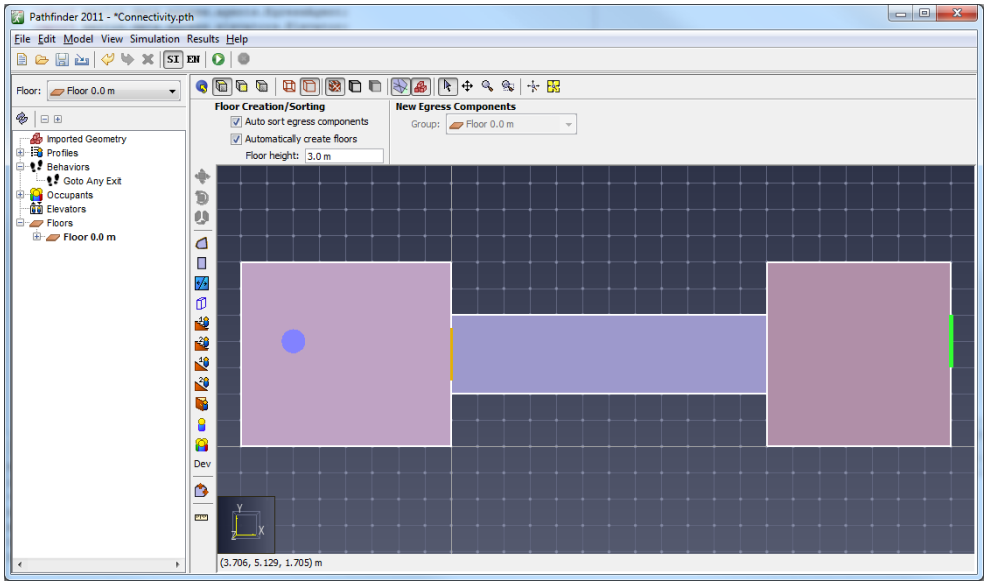
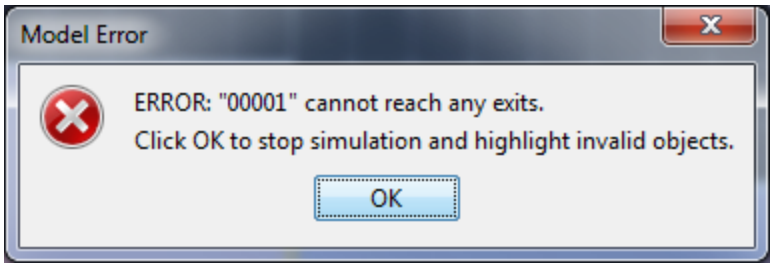


Figure 7-1: Connectivity example

If a user attempts to simulate this model, the following error appears:



This type of error usually indicates that the model is not properly connected. It is telling us that the occupant named “00001” cannot reach any of their assigned exits. This means that either a connecting door is missing along their route to the exit or some other problem has occurred with the geometry. We need to determine why the occupant cannot reach the exit on the right side of the model. To do so, we right-click the room containing the occupant, and click **Select Connected Components** from the right-click menu. The following dialog is shown:

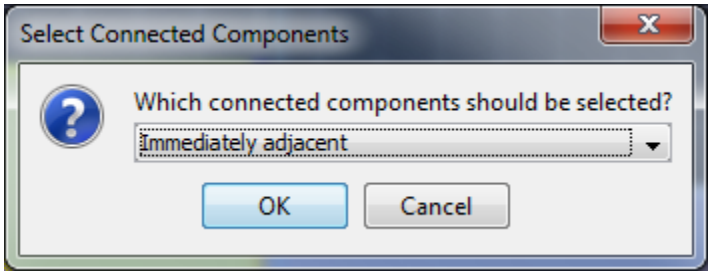


Figure 7-2: Select Connected Components dialog

We want to trace model connectivity as far as possible to determine where we might be missing a door or other connection, so from the drop-down box in the Select Connected Components dialog, we choose **Entire graph** and click **OK**. Pathfinder will highlight the entire graph of components touching the initial selection as shown in Figure 7-3.

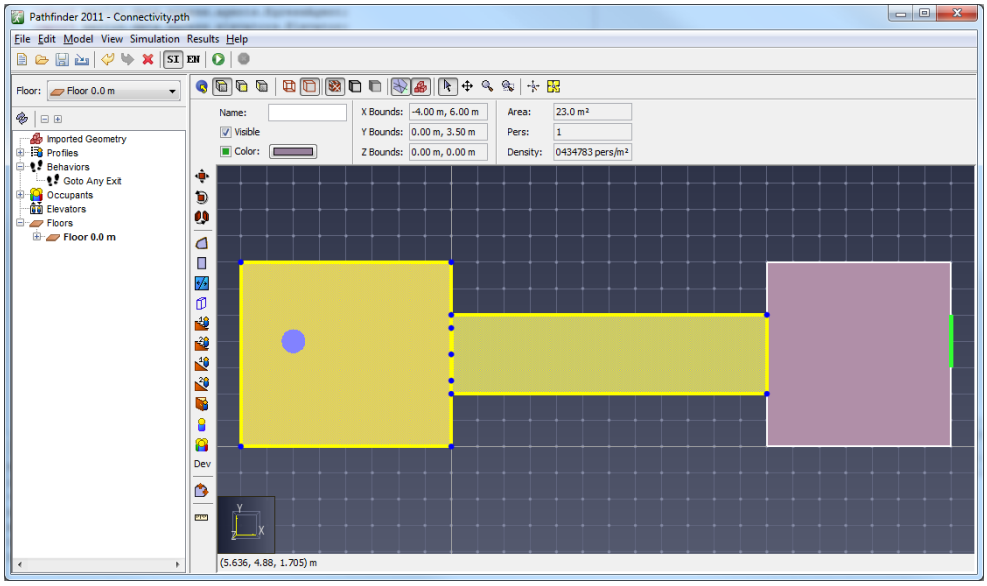


Figure 7-3: Selecting an entire graph of connected components

From the selection, we can see that the hallway does not connect to the far-right room. Upon further inspection, we can see that we do not have a door between the hallway and the room. We can add a door to fix the problem.

From the Select Connected Components dialog, we could have also chosen to select only immediately adjacent components. This would have only selected the original room and the door between the room and the hallway. This allows finer inspection of connectivity in a highly connected model.

Checking in-use Objects

In Pathfinder, there are several objects that can refer to other objects in the model. For instance, an occupant can refer to both a profile and a behavior, a behavior can refer to exits, the goto elevators action can refer to elevators, etc. There are times when it is useful to know what objects are referring to another. For instance, it may be important to know which occupants are using a particular behavior. To do this, right click the behavior (or other in-use object), and from the right-click menu, click **Select Referencing Objects**. This will highlight all objects currently using that behavior.

Warnings and Errors

Pathfinder shows warnings and errors in the Navigation Tree to help debug potential errors in the simulation. These warnings appear as a small exclamation mark imposed over an occupant or record, . By hovering the mouse over the warning, a more detailed description of the problem can be determined. For convenience, if an object has problems, the warning marker is propagated up through the navigation tree to its ancestor groups in order to quickly identify problems in the model. Some common warnings include:

- Warning if stairs are not connected to rooms at both ends.
- Warning if stairs or doors overlap along an edge.

- Warning if stairs or doors create a non-manifold topology (e.g. a stair connects to the inside of a room rather than an outer wall or connects to two rooms on one of its ends).

Problematic objects can be quickly selected by right clicking a group in the tree that has a warning or error icon, and selecting either **Select Errors** or **Select Warnings** from the right-click menu. In addition, if the warning on a component indicates that it interferes or overlaps with other components, the objects with which it interferes can be quickly selecting by right-clicking the object with the warning and selecting **Select conflicting components** from the right-click menu.

As a general note, if the warning “Edge is adjacent to more than 2 triangles” appears when simulating, an option to click **Cancel** appears to highlight the navigation components causing the warning.

8. Simulating

Parameters

The Simulation Parameters dialog provides a way to control certain features of the simulation, as well as provide some default values.

Time Parameters

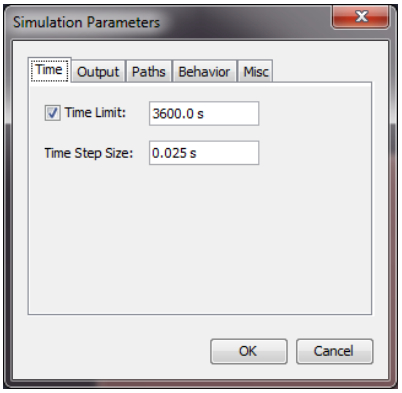


Figure 8-1: The Time tab of the Simulation Parameters dialog

The **Time** tab provides the following options:

- **Time Limit:** can be used to automatically stop the simulation after a set simulation time.
- **Time Step Size:** controls the resolution of simulation time steps. Increase the time step size to speed up simulations, reduce the time step size to ensure simulation accuracy.

Output Parameters

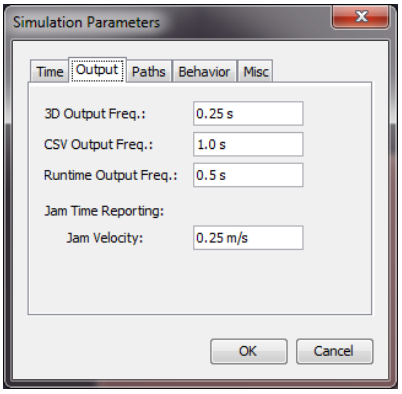


Figure 8-2: The Output tab of the Simulation Parameters dialog

The **Output** tab provides the following options:

- **3D Output Freq:** controls the time between 3D output file updates. Increasing this value causes data to be written less often, leading to less disk usage and run faster simulations (no file write

delay), but can produce a misleading 3D results visualization. In the 3D results visualization, occupants will move in a straight line between two data points - if the two points are far apart in time, an occupant might appear to pass through an obstruction when it actually navigated properly.

- **CSV Output Freq:** controls the time between CSV output file updates. Increasing the value causes fewer rows to appear in the resulting CSV files (i.e. lower data resolution). This option has a negligible effect on simulation performance or disk usage, but can affect performance in the 2D results.
- **Runtime Output Freq:** controls the time between simulation status updates in the Run Simulation dialog. This option has a negligible effect on simulation performance or disk usage.
- **Jam Velocity:** controls the speed threshold at which occupants are recorded as being jammed in the **Occupants** output file.

The **Paths** tab provides the following options:

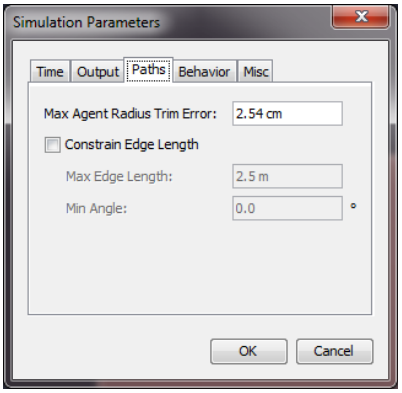


Figure 8-3: The Paths tab of the Simulation Parameters dialog

- **Max Agent Radius Trim Error**: this parameter affects how accurately occupants can navigate through tight spaces when the occupants in the simulation have varying sizes. The larger this value is the less likely an occupant is to navigate through a space that has a width close to their body diameter. With larger values, however, the simulation will consume less memory and start faster (sometimes much faster if every occupant has a different size). Each occupant is guaranteed to be able to fit through a space with width equal to the occupant's diameter plus twice this value.
- **Constrain Edge Length:** controls the triangulation algorithm used to convert the rooms, stairways, etc. into the triangulated mesh used by the simulator. By default, Pathfinder attempts to generate the fewest and largest possible triangles and this approach works well with Pathfinder's search algorithms. However, in some situations, "well behaved" triangles can be useful (e.g. to prevent extremely long, thin triangles). This value can be used to "fatten" these triangles. The **Max Edge Length** parameter controls the maximum length of any single edge on a room boundary, and the **Min Angle** criteria prevents the system from using triangles

with any very small (i.e. thin) angles. Using **Min Angle** values greater than 30 degrees can cause Pathfinder to "freeze" when it attempts to generate a simulation input file.

Behavior Parameters

The **Behavior** tab allows you to set options for Pathfinder's two primary simulation modes: *SFPE* and *Steering*. To select a simulation mode, choose **SFPE** or **Steering** from the **Behavior Mode** drop-down box.

SFPE Mode Parameters

The SFPE mode uses the set of assumptions presented in the *Engineering Guide to Human Behavior in Fire* (SFPE, 2003) and can give answers extremely similar to these hand calculations, depending on selected assumptions. In SFPE simulations, the mechanism that controls simulation movement is the door queue. The SFPE mode uses a simple set of assumptions and usually completes much faster than a comparable steering mode simulation in terms of CPU time.

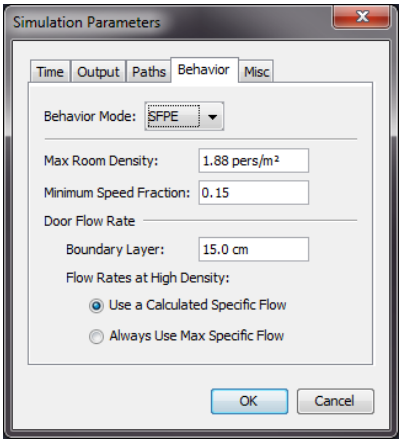


Figure 8-4: SFPE Behavior tab of the Simulation Parameters dialog

The **SFPE** mode supports the following options:

- **Max Room Density:** controls the density at which doors will no longer admit occupants into a room. Using an artificially low value for this number will give faster evacuation times. Using a higher number 3.6 - 3.8 can cause extremely slow evacuation times. Using values above 3.8 pers/m^2 can cause the simulation to become stuck due to the density dependent velocity calculation.
- **Door Flow Rate->Boundary Layer:** This value is subtracted from both sides of a door to calculate the effective door width, controlling the flow rate equation. For example, with a **Door Boundary Layer** setting of 150 mm, a 1.0 m door would be reduced to a 0.7 m opening giving an F_{s_max} of (1.32 pers/s-m * 0.7 m) = 0.924 pers/s.
- **Door Flow Rate->Flow Rates at High Density:** controls how specific flow for doors is calculated. Specific flow is a measure of occupants per unit of time per unit of effective width. For each door, the specific flow is multiplied by the effective door width to calculate the door flowrate in occupants per unit of time.

- **Use a Calculated Specific Flow**: specific flow is calculated as a function of room density for a room adjacent to the door. In cases of counterflow, the room with the higher density controls the combined specific flow of the door.
- **Always Use Max Specific Flow**: Allows the door to flow at the optimum density.

- **Minimum Speed Fraction** can be used to set a lower limit on occupant speed. Setting this value too low can cause evacuation times to increase significantly in cases of high initial loading of rooms or if the **Max Room Density** is set very high.

Steering Mode Parameters

The Steering mode is more dependent on collision avoidance and occupant interaction for the final answer and often gives answers more similar to experimental data than the SFPE mode (i.e. steering mode often reports faster evacuation times). Door queues are not explicitly used in Steering mode, though they do form naturally.

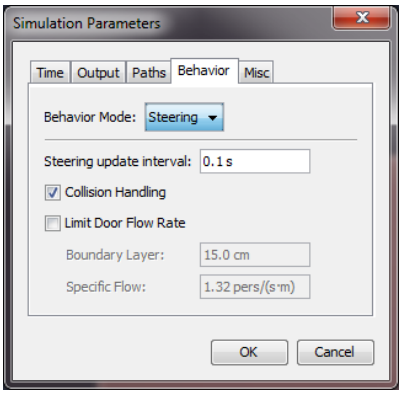


Figure 8-5: The Steering Behavior tab of the Simulation Parameters dialog

The **Steering** mode supports the following options:

- **Steering update interval:** specifies how often (in simulation time) to update the steering calculation. This could also be considered to be the cognitive response time of each occupant. The higher this number, the faster the simulation will run, as long as the simulation time step is less, but the poorer the decision making skills of each occupant will be.
- **Collision Handling:** controls whether occupants avoid one another and can collide with each other.
- **Limit Door Flow Rate:** When checked, this imposes a maximum flowrate on doors unless they have it explicitly turned off (see Door properties on page 26). The flowrate for each door is calculated from the **Boundary Layer** and **Specific Flow**, similarly to SFPE Mode. The difference between Steering and SFPE Modes is that Steering does not allow flowrates to be based off room density.

Starting and Managing a Simulation

To run a simulation: on the Model menu, click Run Simulation... ( ). The simulation will begin and the **Run Simulation** dialog (shown in Figure 8-6) will appear.

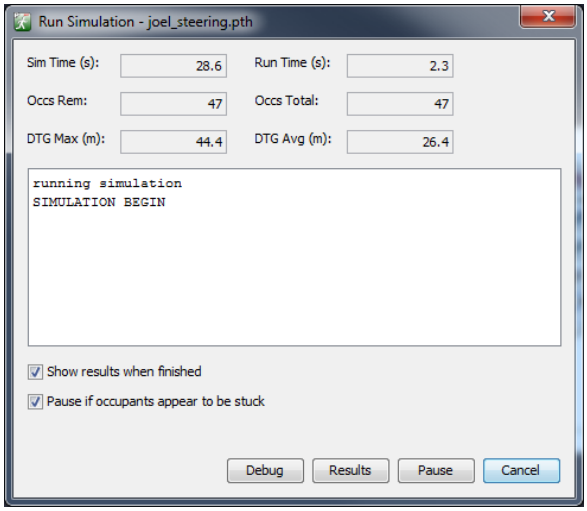


Figure 8-6: The Run Simulation dialog showing a partially complete simulation.

In this dialog, the abbreviation *DTG* stands for distance to goal. The maximum distance to goal represents distance to goal for the occupant farthest from its goal. The average distance to goal is the average of all occupants' distances to their respective goals.

The **Debug** button launches a runtime visualization that shows the progress of the simulation as it is taking place. This function is different from the **Results** button which launches the 3D visualization view for simulation results.

A simulation can also be paused, resumed, and cancelled at any time.

Simulating via command-line

Simulations can also be run through the command-line without loading the user interface. To do so, open a command prompt. Run "testsim.bat" located in the Pathfinder install folder. The only argument is the input .txt file generated by Pathfinder. Pathfinder automatically creates this file when performing a simulation through the user interface. The input file can be manually created in the user interface by going to the **File** menu and selecting **Save Simulator Input…**. When running a simulation this way, it may also be desirable to manually create the geometry file for visualization. To do this, from the **File** menu choose **Save Imported Geometry File…**.

NOTE: Running a simulation through the command line will not provide a management dialog through which the simulation can be paused and resumed.

Stopping and Resuming a Simulation

When running a simulation, there is the option to pause and resume, but this requires Pathfinder to be running the entire time. Sometimes the need arises to be able to stop a simulation and resume later between Pathfinder sessions, such as if the computer needs to be restarted after installing a system update.

To stop a simulation, press **Cancel** during a simulation in the simulation dialog. Pathfinder will ask the user if a snapshot should be made. Before selecting **Yes**, make note of the simulation time. After choosing **Yes**, a snapshot file is written into the same directory as the result with a filename similar to the following:

untitled_XXXXX.XXX.snapshot

where the X's are replaced with the simulation time at which the simulation was stopped. The snapshot file can later be resumed. To resume it, go to the **Simulation** menu, and select **Resume Simulation…** Then select the snapshot file. The simulation will resume from the time it was stopped. Note that the Pathfinder file from which the simulation originated should be open to resume a simulation.

Stuck Occupants

Sometimes occupants become "stuck" preventing a proper simulation run. There can be many causes of this problem and we do everything we can to prevent it, but it does happen. If occupants are becoming stuck in tight spaces, consider the following steps to resolve the problem:

1. Alter the navigation mesh in the area where occupants are becoming stuck. This can be especially useful if the area was originally extracted from imported geometry. Sometimes simply re-drawing the area using the drawing tools can fix the issue.
2. Slightly increase the occupants' **Comfort Distance** in their profile.
3. Slightly reduce the occupants' **Reduction Factor** in their profile.
4. Switch to the SFPE simulation mode. This mode uses the simplest set of assumptions and is not as susceptible to the geometric and movement irregularities that can cause occupants to become stuck.

In addition, please contact Thunderhead Engineering for support at mailto:support@thunderheadeng.com?subject=Stuck Pathfinder Model.

9. Results

Summary Report

The summary report file contains information about the simulation geometry, simulation performance, and usage information for each room, stairway, and door. Figure 9-1 shows a portion of an example summary report file.

```
***SUMMARY***SUMMARY***SUMMARY***SUMMARY***SUMMARY***

Simulation:         MultiFloorStairwell
Mode:               SFPE (Basic)
Total Occupants:    125
Exit Times (s):
  Min:              3.2
  Max:              176.7
  Average:          85.0
  StdDev:           53.7

[Components] All:   28
[Components] Doors: 14
Triangles:          66
Startup Time:       0.0s
CPU Time:           2.4s

            ROOM/DOOR   FIRST IN (s)   LAST OUT (s)   TOTAL USE (pers)   FLOW AVG. (pers/s)
--------------------  ----------  ----------  ----------  ---------
               Room01         0.0        26.7          25
               Room02         0.6       176.7         125
               Door00         3.2       176.7         125         0.7
               Door01         0.6        26.7          25         1.0
              Stair01         3.6       171.3         100
      Stair01 door 1         10.3       171.3         100         0.6
      Stair01 door 2          3.6       152.8         100         0.7
               Room02         2.6       152.8         100
               Room01         0.0        28.7          25
               Door03         2.6        28.7          25         1.0
              Stair02         2.8       127.5          75
      Stair02 door 1          9.6       127.5          75         0.6
      Stair02 door 2          2.8       106.1          75         0.7
               Room01         0.0        26.7          25
               Room02         0.6       106.1          75
               Door04         0.6        26.7          25         1.0
              Stair03         3.6        84.8          50
      Stair03 door 1         10.3        84.8          50         0.7
      Stair03 door 2          3.6        64.2          50         0.8
               Room02         2.6        64.2          50
               Room01         0.0        28.7          25
              Stair04         2.8        36.3          25
      Stair04 door 1          9.6        36.3          25         0.9
      Stair04 door 2          2.8        28.7          25         1.0
               Door07         2.6        28.7          25         1.0
               Room02         0.6        28.7          25
               Room01         0.0        26.7          25
               Door08         0.6        26.7          25         1.0
--------------------  ----------  ----------  ----------  ---------
```

Figure 9-1: Listing for an example summary report file.

This file is saved in the simulation directory and given the name *name*_summary.txt (where *name* is the name of your saved PTH file). To view it, under the **Results** menu choose **Show Summary File**. The first section shows the mode the simulation was run in, the total number of occupants, and statistics on the evacuation time. It also shows some information about the mesh, including the number of triangles and the doors. This information can be useful when considering the complexity of a simulation from the standpoint of the simulator.

The table gives a listing of each component (doors, rooms, and stairs) in the simulation. For each component, the FIRST IN column shows the simulation time when the first occupant entered that component. LAST OUT shows the simulation time when the last occupant exited that component. The TOTAL USE column shows how many times a component was entered by occupants. For doors that served more than 1 occupant, the FLOW AVG. column shows the result of dividing the total use by the amount of time the room was in use (LAST OUT - FIRST IN).

Door History

The door history file (*name*_doors.csv, where *name* is the name of your saved PTH file) provides results data for doors. Each row represents a different time step, and the columns are as follows:

- **time(s):** The output time for this data row. The frequency of output is controlled by the **CSV Output Freq.** box in the **Simulation Parameters** dialog.
- **Remaining (total):** The number of occupants remaining in the simulation.
- **Exited (total):** The number of occupants that have successfully passed through an exit door (leaving the simulation).
- ***doorname* width(m):** The total width of the specified door.
- ***doorname* total boundary(m):** The total boundary layer of the specified door.
- ***doorname* [*{+,-}{X,Y}*]:** The number of occupants who have passed through door *doorname* since the previous output in the specified direction. For columns with no direction specification, this is the total number of occupants to pass through the door in both directions since the previous output.
- ***doorname* (Q):** The number of occupants who are waiting in the queue to pass through door *doorname* at the current time. This only includes occupants who have actually reached the door and are waiting to enter. Occupants that are stacked up waiting to reach a door will not be counted. This value is only meaningful in SFPE mode.

This file is used to display door flowrate, specific flow, and usage history in Pathfinder.

Door Flowrate and Specific Flow

To view door flowrate or specific flow, click **View Door Flow Rates...** on the **Results** menu. This opens a time history plot for doors as in Figure 9-2. This plot shows data from the door history file. In the left portion of the window is a list of the doors, and on the right is a graph of the data.

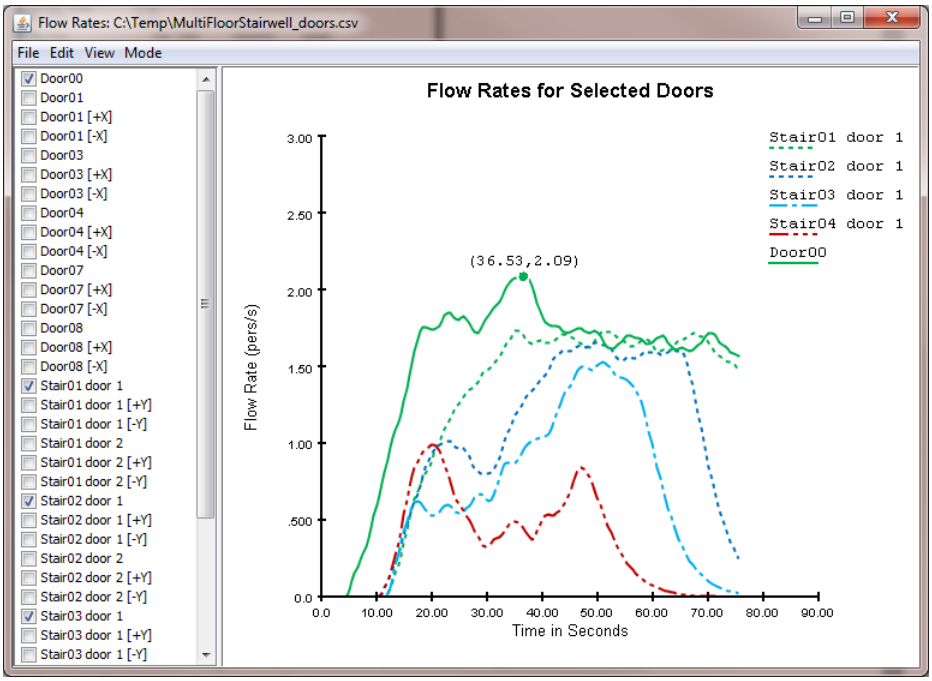


Figure 9-2: A time history plot for door flow rates

By default, door flow rates are shown. Alternatively, on the **Mode** menu, choose **Specific Flow** to view door specific flow.

There are three filtering modes for presenting the flowrate that are selectable through the **View** menu:

- **Raw:** this provides raw flowrate, which is simply *num_occs*/*dt*, where *num_occs* is the number of occupants to pass through the door in an output time step, and *dt* is the output time step.
- **Low-pass Filter:** the raw flowrate is filtered with a bi-quad low-pass filter with a user-specified cutoff frequency. This is the default filter, and the default cut-off frequency is .05 Hz. Lower cut-off frequencies produce smoother graphs.
- **Moving-average Filter:** the raw flowrate is averaged over a user-specified period.

Door Usage

To view door usage, open the **Door Flowrate Rates** dialog, and on the **Mode** menu, select **Occupant Counts**. This shows the number of occupants that use a particular door in each output time step.

Alternatively, cumulative totals can be viewed by selecting **Cumulative Occupant Counts**. This shows the total number of occupants to use the door up until that time.

Room History

The room history file (*name*_rooms.csv, where *name* is the name of your saved PTH file) provides the following information columns in each row:

- **time(s):** The output time for this data row. The frequency of output is controlled by the **CSV Output Freq.** box in the **Simulation Parameters** dialog.

- **Remaining (Total):** The number of occupants still in the simulation.
- **Exited (Total):** The number of occupants that have successfully passed through an exit door (leaving the simulation).
- ***roomname:*** The number of occupants present in room (or stairway) *roomname* at the current time.

To display this data as a time history plot, click **View Room Usage...** on the results menu.

Occupants Summary

The occupants summary file (*name*_occupants.csv, where *name* is the name of your saved PTH file) provides statistics about each occupant in the simulation. Each row of the CSV represents one occupant and provides the following:

- **id**: A unique integer identifier for the occupant assigned by the simulator.
- **name:** The name given to the occupant in the user interface.
- **exit time:** The time at which the occupant exited the simulation.
- **active time:** The amount of time the occupant was actively seeking an exit.
- **jam time total(s):** The total amount of time the occupant spent moving at less than the **Jam Velocity** as specified in the simulation parameters (see Output Parameters on page 83).
- **jam time max continuous(s)**: The maximum continuous amount of time the occupant spent at less than the **Jam Velocity**.

Occupant History

For each occupant whose profile has CSV data output enabled (see Profiles on page 50), an occupant history file is created with the file name, *name*_occupant_*id*_*occname*.csv, where *name* is the name of the saved PTH file, *id* is the integer id of the occupant assigned by the simulator, and *occname* is the name of the occupant specified in the user interface. Each row in an occupant history file shows the following data for a particular time:

- **t(s)**: The output time for this data row. The frequency of output is controlled by the **CSV Output Freq.** box in the **Simulation Parameters** dialog.
- **id:** The integer identifier of the occupant assigned by the simulator.
- **name**: The name assigned to the occupant in the user interface.
- **active:** Whether the occupant is actively seeking an exit (1 if they are seeking an exit and 0 if not).
- **x(m), y(m), z(m):** The 3D location of the occupant.
- **v(m/s):** The velocity of the occupant.
- **distance(m):** The total distance the occupant has travelled.
- **location:** The occupant's current room.

3D Results

Pathfinder provides a real-time output visualization program for viewing 3D results. It operates much like a video player in that it allows users to play, pause, stop, slide, and speed up and down time. It is completely 3D and allows users to navigate through a model.

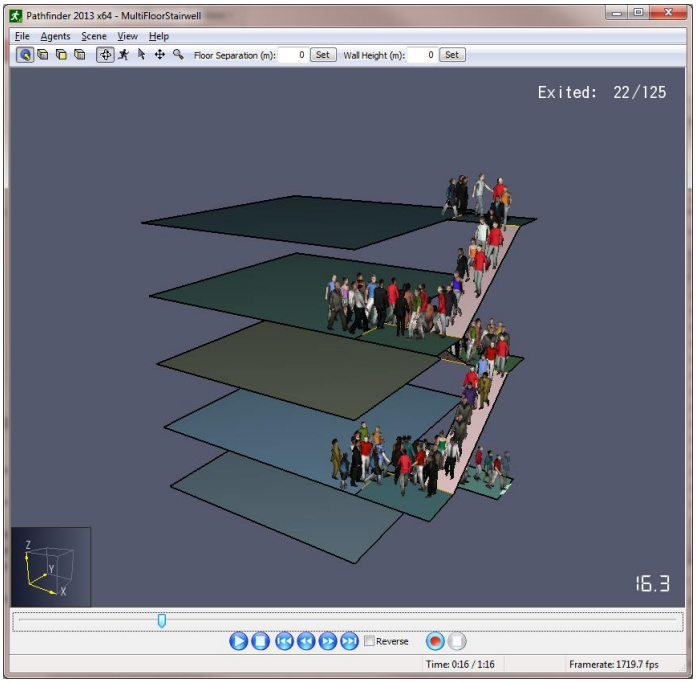


Figure 9-3: 3D Results for the multi-floor-stairwell example problem

Navigating through a model

Once a model is loaded, the navigation tools work similarly to those found in the preprocessor. The user can rotate the model ( ), zoom in and out ( ), and roam ( ).

In the 3D results, the roam tool provides additional capabilities for navigating through the model.

- **Course Positioning:** As in the preprocessor, the roam tool allows the camera to be quickly moved throughout the model at the expense of precision. To move the camera in this way in the XY plane, hold Ctrl while dragging the left mouse button, or drag the middle mouse button. To move the camera along the Z axis, hold Alt while dragging the left mouse button or drag the right mouse button.
- **Precise Positioning:** When the camera is roughly in the correct spot, it can be moved to a more precise location by using the keys, 'w' and 's' to move the camera forward and backward along the viewing direction, and the keys 'a' and 'd' to move the camera left and right, respectively. In addition, 'Space' moves the camera up along the Z-axis, and 'c' moves the camera down. Pressing these keys moves the camera at a fixed speed. Holding 'Shift' while pressing them doubles the speed.
- **Smooth Positioning:** The roam tool can also be used to smoothly animate the camera to any location. To do so, press and release the middle mouse button. The cursor will disappear, and

the tool will enter **Roam Mode**. In this mode, moving the mouse will pivot the camera about its location. Pressing and dragging the left mouse button will move the camera in the XY plane. The further the mouse is moved from its button press, the faster the camera will move. This can simulate the effect of accelerating the camera. Doing the same with the middle mouse button will cause the camera to move forward/backward in the XY plane with changes along the Y mouse axis and turn left/right with changes along the X mouse axis. Pressing and dragging the right mouse button will move the camera along the Z axis in the same manner. To exit **Roam Mode**, press and release the middle mouse button again or press Esc on the keyboard.

At any time, press Ctrl+R to reset the camera view.

Displaying geometry input

If a DXF, FDS, PyroSim, or background image file has been imported in a Pathfinder simulation, this geometry will be shown by default in the 3D results. If not, the navigation geometry used by the simulator will be displayed, which only contains walking space, doors, stairs, and exits. These two geometry sets can be viewed in combination or individually through the **Scene** menu:

- **Show Nav Mesh over Imported Geom** – this shows the navigation mesh and the imported geometry with the navigation mesh on top. This is useful to see how the navigation mesh lines up with the imported model.
- **Show Imported Geom over Nav Mesh** – like the first option, this shows the navigation mesh and the imported geometry together but with the imported geometry on top. This display is useful if the imported geometry is the most interesting geometry to display but the navigation mesh fills in some missing pieces from the imported geometry, such as stairs or elevators.
- **Show Nav Mesh Only** – this shows only the navigation mesh.
- **Show Imported Geom Only** – this shows only the imported geometry.

Displaying occupants

Occupants can be displayed in several ways. They can be shown as simple shapes, as realistic people, or as the artist's wood mannequin. All display options are found under the **Agents** menu.

The first way to display occupants is as simple shapes, including disks and cylinders. Because of the simplicity, this is the fastest way to view agents (for loading and on some graphics cards, rendering speed), but it is the least realistic and contains little animation.

Another way to display occupants is as realistic people. The avatar selected in the pre-processor determines which avatar is shown in the 3D results. Displayed this way, occupants may render slower on some graphics cards and will take longer to load, but the scene will appear much more realistic and should still be fast enough to display hundreds to thousands of occupants depending on the processor speed and graphics card.

The last way to display occupants is as the artist's mannequin. Generally, this is faster than showing people and is completely generic, contains animation, and still provides a degree of realism above simple shapes. It does not, however, represent colors set on the occupants.

Selecting Occupants

Occupants can be selected for tracking purposes by clicking on the occupant. When selected, the occupant and their path (if displayed) will turn yellow. Multiple occupants can be selected by holding CTRL when clicking them. They can be de-selected by clicking anywhere else in the model.

Viewing multi-floor problems

Some models are composed of several floors, causing each floor to obscure the one below it as shown in Figure 9-5 (a). Pathfinder provides a variety of options for viewing the model such that the occupants inside can be easily observed. One option is to view the model with the floors stacked vertically but separated by a set amount. Another option is to view the model with the floors laid out in the XY plane so that they can all be viewed from overhead. Both options allow a wall clip-height to be specified.

To use any of the layout options, floors must first be defined. If the floors were defined in the pre-processor, they will carry through to the 3D visualization and no additional input is needed. If they have not been defined or they need to be modified, this can be done by opening the **Scene** menu and selecting **Edit Floors…**. A dialog will appear that shows a list of the Z locations of the floors as shown in Figure 9-4. From this list, floor locations can be added, removed, and modified. Once OK is pressed, the 3D results viewer will partition the geometry among the floors such that the bottom floor contains all geometry from -∞ to the next floor's location, the top floor contains all geometry from its location to +∞, and every other floor contains geometry from its location to the next floor's location.

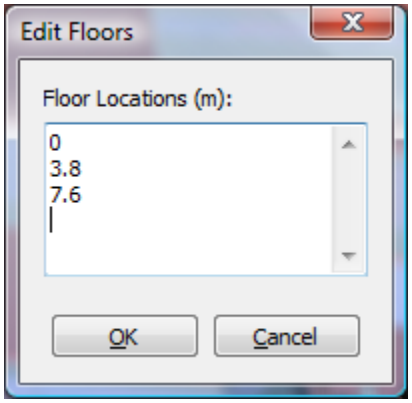


Figure 9-4: Floor location dialog for 3D results

Once the floors are defined, wall clipping can be applied to shorten the walls yet maintain a sense of obstruction placement as shown in Figure 9-5 (b). As shown in this figure, more of the model can be seen than in Figure 9-5 (a). The wall height can be set in the toolbar above the 3D view. If this value is set above 0, the geometry on every floor will be clipped above the floor's Z location plus the wall height. If the value is set to 0, the full wall height is used.

To stack floors vertically along the Z axis, open the **Scene** menu and select **Layout Floors Vertically**. When viewed in this manner, floors will appear naturally such that each floor is stacked on top of the one below it. The floors can then be separated by empty space to more easily view inside the model as shown in Figure 9-5 (c). This **Floor Separation** can be set in the toolbar above the 3D view.

To layout floors in the XY plane, open the **Scene** menu and select **Layout Floors Horizontally**. The floors will be laid out in a square grid and in one plane. They will appear in order from left to right and top to bottom such that the lowest floor is in the upper left corner and the highest floor is in the lower right corner. When laid out in this manner, the **Floor Separation** will affect the gap between floors in both the X and Y directions. An example of this type of layout is shown in Figure 9-5 (d). By default, labels appear above each floor so the floors can be easily identified, but these labels can be hidden by opening the **View** menu and deselecting **Show Labels**.

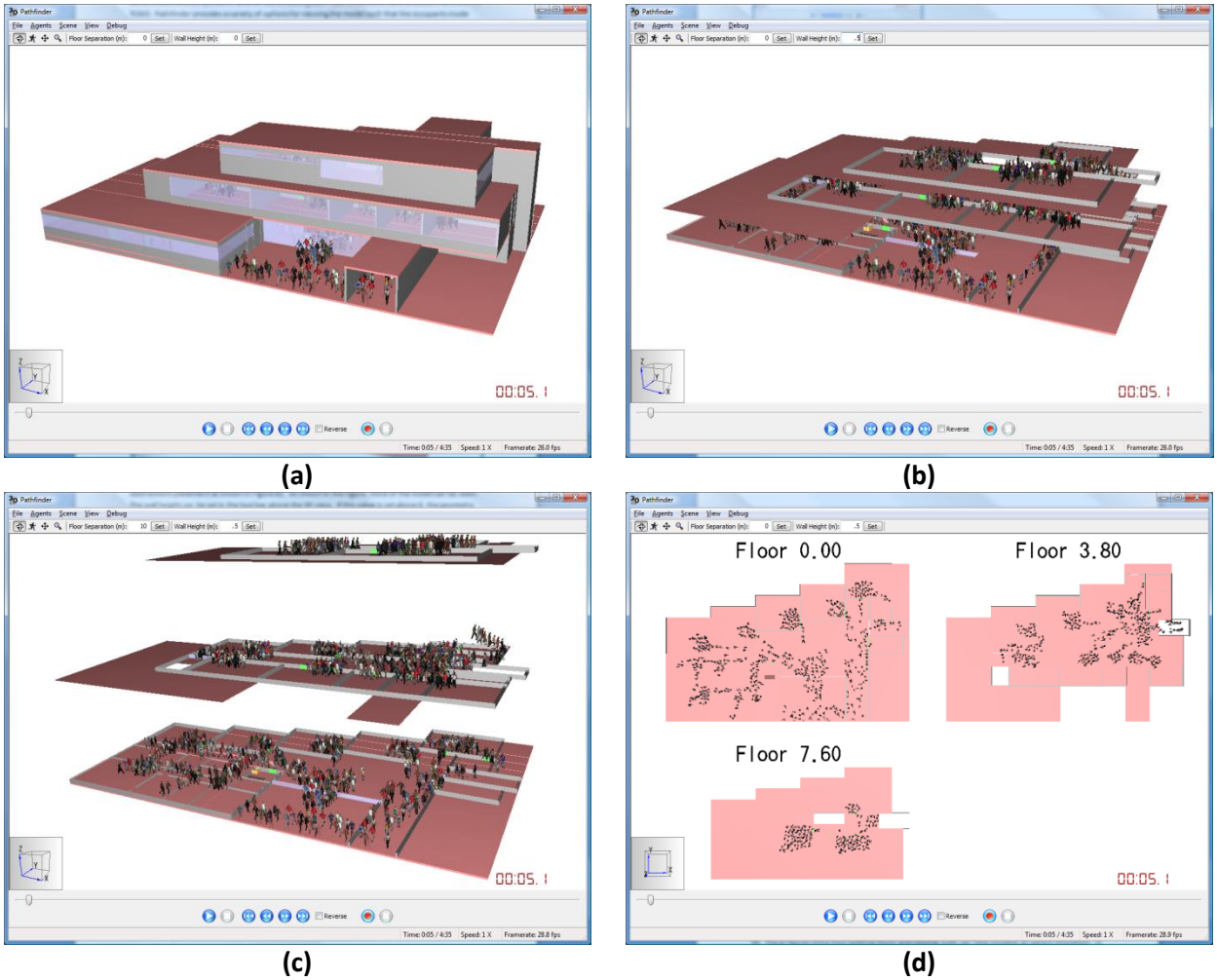


Figure 9-5: Multi-floor layout options in 3D results (FDS model courtesy of Andreas Niggemeyer)

Animation playback

To play back animation, press the play button ( ). As the animation is playing, the time slider will move, and the animation can be paused ( ) or stopped ( ). The buttons ( ) and ( ) will move the time slider to the beginning and end, respectively. The buttons ( ) and ( ) will slow down and speed up the animation by factors of two.

The status bar at the bottom of the screen shows information about playback. The first section from left to right shows the current playback status. This will display “Playing,” “Paused,” or “Stopped.” The

following section shows the current playback time in minutes. The next section shows the current playback speed. The last section shows the current rendering speed in frames per second – a common speed measurement in computer graphics. This can help determine how well a user's computer handles 3D playback. Some ways to increase this are discussed in the section, Controlling drawing detail/speed.

Refreshing Results

Results can be viewed as a simulation is running, but they will only be current to the last load or refresh. To refresh the results at any time, press *F5* on the keyboard or select **Refresh Results** in the **File** menu. The current playback time will be restored after refreshing the results, making the refresh as seamless as possible. If the results are viewed before the simulation is finished, upon completion the results will automatically be refreshed and brought to the front of the screen.

Viewing occupant paths

One option that can help visualize flow in an evacuation is to turn on occupant paths. This can be enabled by checking **Show Occupant Paths** in the **View** menu. When turned on, each occupant's path will be drawn from t=0 to the current playing time.

Controlling drawing detail/speed

There are several options controlling the drawing detail and ultimately speed in the 3D results, including level-of-detail rendering, hardware versus software rendering, and vertex buffers.

One way to vastly increase speed while only somewhat sacrificing quality is to enable view-dependent level-of-detail rendering. This can be enabled by checking **Enable Level of Detail** under the **View** menu. The idea behind this option is that objects far from the camera require less detail to portray the object realistically than for those close to the camera.

Another improvement for many modern graphics cards is to animate the occupants in hardware. To do this, select **Animate Characters in Hardware** under the **View** menu. For these graphics cards, this, coupled with vertex buffers can vastly improve performance. Some older graphics cards, however, will support it but will actually render slower when using it. It is recommended that the user play with this option and observe the difference in frame rate (shown at the bottom right in the status bar). Recommended graphics cards include the Nvidia Geforce (GTX, especially) and AMD Radeon HD series. These will work best with the hardware animation option.

The last speed option available is using vertex buffers, which is enabled by checking **Enable Vertex Buffers** under the view menu. Hardware vertex buffers allow geometric data to be stored in video memory to reduce data that must be sent over the graphics bus from main memory, which can increase rendering speed. We have found, however, that integrated Intel video cards often have trouble with this option, so it may need to be disabled for these cards.

The two options, **Animate Characters in Hardware** and **Enable Vertex Buffers**, when combined tend to provide the most benefit on newer graphics hardware.

Showing Saved Views

If any views were created in the pre-processor (see Views on page 66), they will appear in a navigation view on the left side of the window. The navigation view can be hidden by going to the **View** menu and de-selecting **Show Navigation View**.

To show one of the saved views, double-click the desired view in the navigation view. This will automatically switch to the perspective camera and show the view.

Showing Camera Tours

If any camera tours were created in the pre-processor, they will be available in the navigation view on the left side of the window. To begin playback of a camera tour, either double-click it in the navigation view or select it and click the **Show Camera Feed** button ( ) in the main toolbar.

To stop the camera tour, click the **Stop Camera Feed** button ( ) in the main toolbar.

Creating Movies

Movies can be created from the 3D results for playback on a PC using a video player such as Windows Media Player. Creating a movie file allows the results to be distributed to others who do not have the 3D results software that is bundled with Pathfinder. Movies can be created in two different ways. One way is to create a high-quality movie that is rendered offline. The other is to create a lower-quality movie in real-time that allows the user to interact with the scene while the movie is recording.

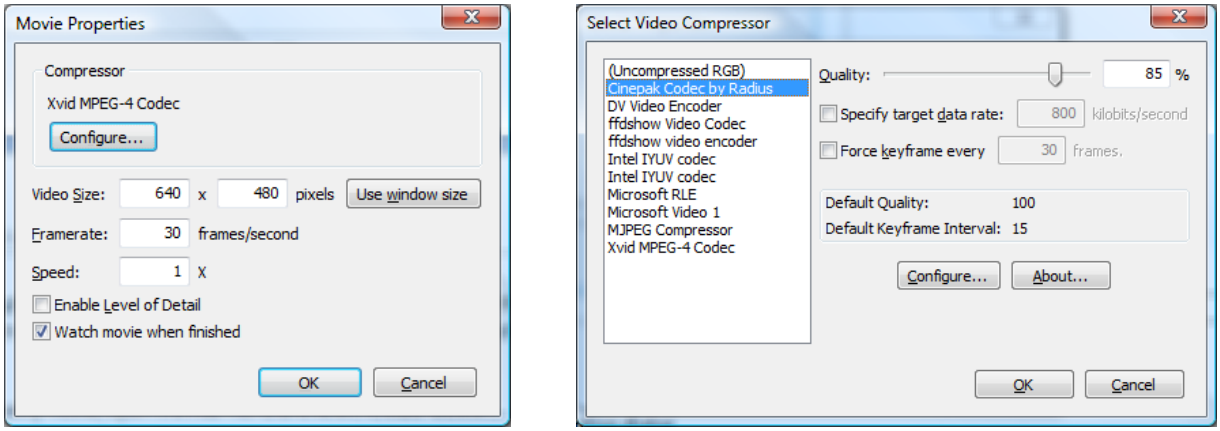
High Quality Movies

High quality movies are useful if a user wants a movie that is unrestricted by size and frame rate. This means that it will play back smoothly at any resolution and with any high-quality settings turned on, regardless of the complexity of the scene. The disadvantage to this type of movie is that the camera movements must be pre-planned using camera tours. Otherwise, the camera will remain static. In addition, the user cannot interact with the scene while the movie is being created (such as changing the selected occupant, turning on and off paths, etc.).

To create a high-quality movie:

1. Activate the desired camera tour to be used for the movie by double-clicking it in the navigation view.
2. In the **File** menu select **Create Movie…**. A file chooser will ask for the file name of the resulting movie. From this file chooser, the type of movie can be specified as an **Audio-Video Interlaced** (AVI) file or a **Windows Media Format** (WMV) file. AVI files can be rendered as *uncompressed* or *compressed*. WMV files can only be compressed. Uncompressed files will be very large but will be of the highest-quality. Compressed files are much smaller in size but with some reduced quality. To create compressed AVI files, video *codecs* must be installed on both the computer that will create the file and the computer that will play back the file. Some popular AVI codecs include Xvid, Divx, and ffdshow. WMV files, however, do not require special codecs to be installed on either the source or destination computers, but they will only work on Windows machines unless special software is installed on another platform.

3. A dialog will show some movie making options as in Figure 9-6 (a). The options are as follows:
   - **Video Size** specifies the resulting video's dimensions. This is only limited by the maximum size of a viewport on the computer's video hardware; with some codecs, however, each dimension must be a multiple of 2 or 4. If you have trouble creating a movie, try making the size a multiple of 4. The **Use window size** button will set the size to the 3D window's current size.
   - **Framerate** determines how smoothly the video will play back. The higher this number is, the smoother the playback at the expense of longer rendering time to create the movie and larger file size. 30 frames/second provides reasonably smooth video playback.
   - **Speed** specifies the speed at which the results will play. 1 X will play the movie at real-time speed, 2 X will play the movie twice real-time, etc.
   - **Enable Level of Detail** specifies that level of detail rendering should be turned on while creating the movie. This will lower the resulting quality, but it will decrease the time to make the movie on scenes with many occupants.
   - **Watch Movie when Finished** specifies that the resulting movie should open in the default video player for the chosen video type when the movie is completed.
   - **Compressor** shows which video codec is currently selected and allows the codec to be chosen and configured. If **Configure…** is pressed, another dialog will show as in Figure 9-6 (b). This dialog will show a list of the codecs currently installed on the computer. Some codecs allow certain properties, such as the **Quality**, **Data Rate**, and **Keyframe interval** to be specified. These properties control the resulting movie's quality, file size, and seeking performance (how well a movie can be skipped around in the video player), respectively. Some codecs do not expose these properties through this dialog, however. To configure these codecs, you must press the additional **Configure…** button if it is enabled.



(a) **(b)**

Figure 9-6: Movie option dialogs

4. Press **OK** to start making the movie. While the movie is being created, a dialog will show the progress. If the movie is cancelled before it is completed, there will still be a valid video; it just will not play the entire results.

Real-time Movies

Movies can also be created in real-time. This means that it will record while the user is interacting with the results and it will record everything the user sees except for the cursor. The disadvantage of this type of movie is that it will play back only as smooth and of the same quality as that which the user who created the movie saw. So if the scene is very complex and the movie creator's computer cannot view the results smoothly, the resulting movie's play back will suffer as well.

To create a movie of this type, press the record button ( ). A file chooser dialog will prompt for the file name of the movie. For real-time movies the only allowable type is WMV. Once the file name is chosen, the codec configuration dialog will be shown as in Figure 9-6 (b). When **OK** is pressed on this dialog, the movie will begin recording. At this point, anything the user sees will be recorded in the movie. Now the user can change the camera angle, start and stop playback, select occupants, change view settings, etc. and the movie will capture everything. At any time, the movie can be paused or stopped. Pausing the movie will stop the recording but leave the movie file open so that recording can be resumed to the same file. This allows the user to perform some action that they do not want to be recorded. To pause movie recording, press the pause movie button ( ). To resume recording, press the resume movie button ( ). Stopping the movie will end recording and disallow any further recording to the file. To stop recording, press the stop movie button ( ).

Showing FDS Results

Pathfinder provides some support for viewing animation results from the Fire Dynamics Simulator (FDS) from NIST. Current support provides the ability to view animated slices overlaid with Pathfinder results.

To load FDS results, first open the desired Pathfinder results. Then, under the **Analysis** menu, select **Load FDS Results…**, and choose a Smokeview file (.smv) that was generated by FDS. The **Navigation View** will open on the left, listing all of the supported FDS output as shown in Figure 9-7.

To unload FDS results, under the **Analysis** menu, choose **Unload FDS Results**.

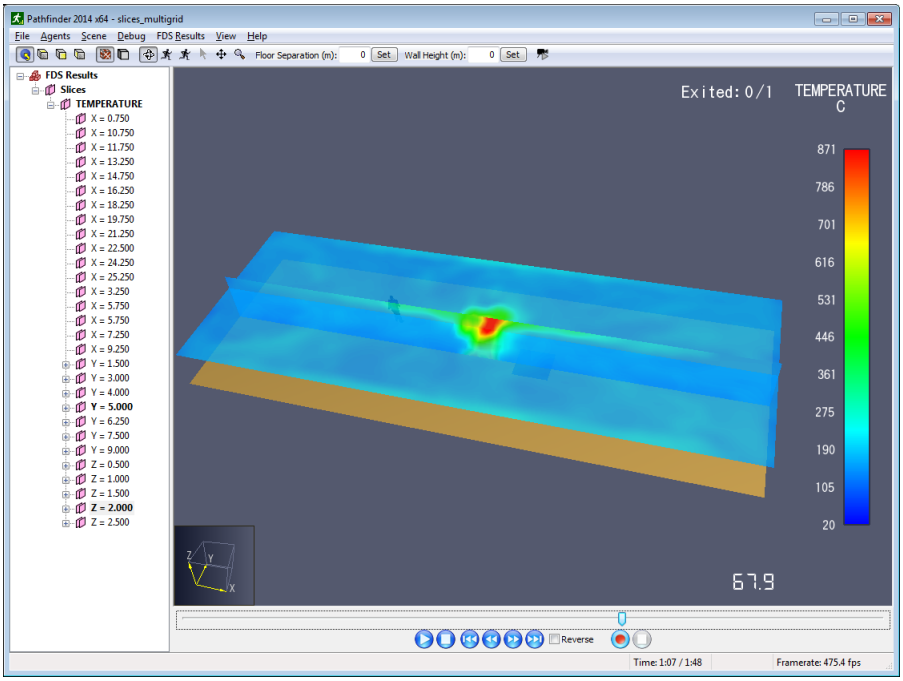


Figure 9-7: FDS Slices

Slices

When FDS results are loaded, the **Navigation View** will show the FDS slices that are available. Initially, all slices are unloaded and must be manually loaded to show them in the results.

To load a slice, either double-click it in the navigation view or right-click it and select **Show**. When loaded, the slice name will appear in bold in the navigation view. Several slices may be loaded at the same time as long as they all show the same quantity, for example TEMPERATURE.

To load all slices of a particular quantity at once, double-click the desired quantity in the navigation view or right-click and choose **Show**.

To unload a slice, double-click a loaded one in the navigation view, or right-click it and select **Hide**. Similarly, all slices may be unloaded by performing this for the currently loaded quantity.

To change the coloring or data range of the slices, under the **FDS Results** menu, choose **Preferences…**. The preferences dialog will open as shown in Figure 9-8.

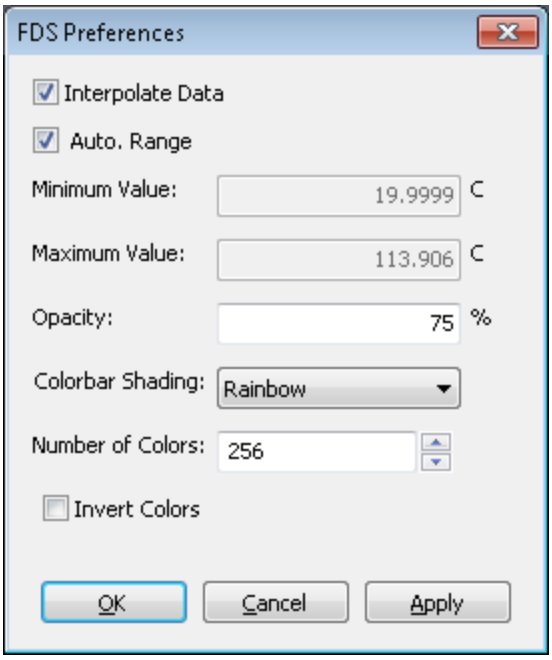


Figure 9-8: FDS preferences dialog

The options are as follows:

- **Interpolate Data**: Interpolates data between frames over time. This allows the animation of the slices to look smooth even when FDS outputs the slice data at large time intervals. Uncheck this to view data exactly as generated by FDS and make slice animation more closely match that in Smokeview.
- **Auto. Range:** Automatically sets the minimum and maximum values based on the maximum range of the loaded slice files (NOTE: These automatic values are actually the middle 98% of values in the results rather than the absolute minimum and maximum).
- **Minimum Value:** If **Auto. Range** is unchecked, this sets the minimum display value.
- **Maximum Value:** If **Auto. Range** is unchecked, this sets the maximum display value.
- **Opacity:** Sets the opacity of the displayed slices. Values approaching 100% make the slices look more solid, where values closer to 0% make the slices look more transparent.
- **Colorbar Shading:** Specifies the color palette to display.
- **Number of Colors:** Species how many colors are available in the palette. This must be between 2 and 256.
- **Invert Colors:** Reverses the color scale.

Occupant Contours/Heat Maps

Pathfinder supports the visualization of dynamic occupant data on the floor areas of the model as shown in Figure 9-9. Referred to as occupant contours or heat maps, this animated data can give a qualitative visualization of the model's performance.

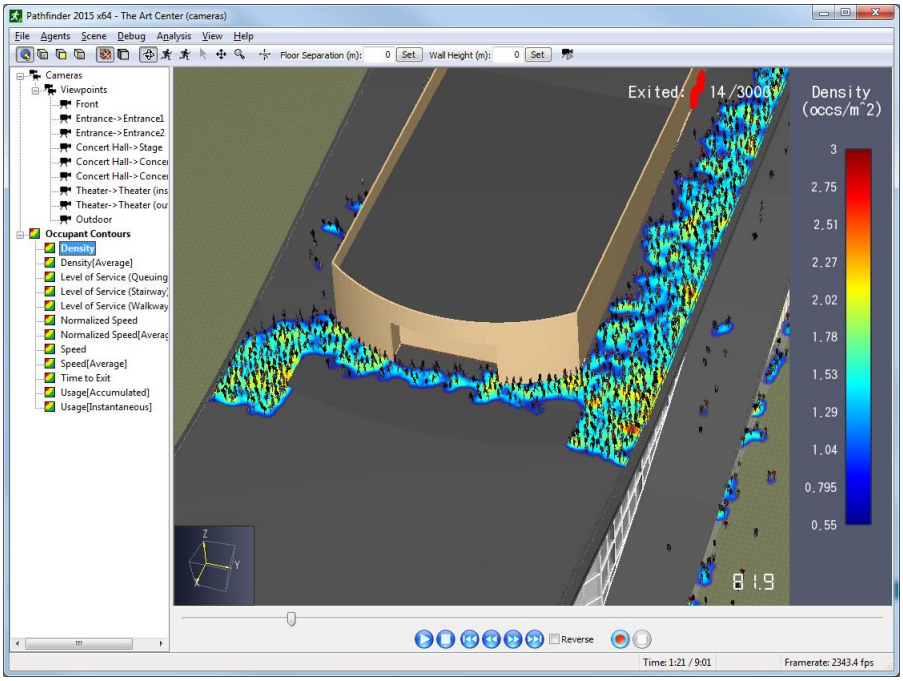


Figure 9-9: Occupant Contours

Pathfinder supports the visualization of several types of occupant contours, including the following:

- **Density:** Shows how densely packed occupants are in occupants/m^2.
- **Level of Service:** Shows Level of Service (Fruin, 1987) defined by surface type, including stairways, walkways, and queuing areas.
- **Speed:** Shows occupant speed.
- **Normalized Speed:** Shows occupant speed normalized against each occupant's maximum speed.
- **Time to Exit:** Shows the amount of time it will take an occupant to exit from each point on the floor given the current playback time.
  NOTE: There will only be data from occupants that have a simulated exit time. If this quantity is viewed before a simulation is complete, the results may be incomplete. The contour will update, however, when results are refreshed.
- **Usage [Instantaneous]:** Shows where occupants currently exist on the floor. This can be used as an alternative to viewing individual occupants, which is useful when viewing large models from afar.
- **Usage [Accumulated]:** Shows a time-integrated version of **Usage**, which allows the user to see how much time is spent on each area of the floor.

In addition to the basic contours, Pathfinder also provides the ability to filter any contour as follows:

- **Average:** Averages the values of the base contour over a trailing time interval (default = 30 s). This can be useful for smoothing data over time and to act as a low-pass filter.
- **Maximum:** Calculates the maximum contour value over a trailing time interval (default = unlimited).

The ability to filter any contour gives the user a great deal of flexibility in visualizing contours. For instance, a contour could be created that displays the maximum average density. The averaging filter would act as a low-pass filter that reduces high-density peaks that may be short lived. The maximum filter would then be used to find the maximum of this smoothed data. This would be created by adding an average filter to the density contour and then adding a maximum filter to this averaging filter.

Adding/Removing Contours

By default, a results file will contain several contours. More can be added or removed at any time. Adding a contour can be done in one of two ways:

- On the **Analysis** menu, select **New occupant contour…** ( ) or
- Right-click the "Occupant Contours" group in the navigation view and select, **New occupant contour…**.

Performing either of these actions will open the **New Occupant Contour** dialog. In this dialog, select the desired contour quantity, and click **OK**. This will create the new contour and show its properties dialog as discussed below.

Contours can be removed at any time by selecting the desired contour in the navigation view, and either pressing DELETE on the keyboard or selecting **Delete…** ( ) from the right-click menu.

Adding Contour Filters

Contour filters can be added at any time in one of two ways:

- On the **Analysis** menu, select **New contour filter…** ( ) or
- In the navigation menu, right-click the contour to filter and select **Add filter…**.

This will show the **New Filtered Occupant Contour** dialog as shown in Figure 9-10. Select the desired properties and press **OK** to create the filter and show its properties dialog.

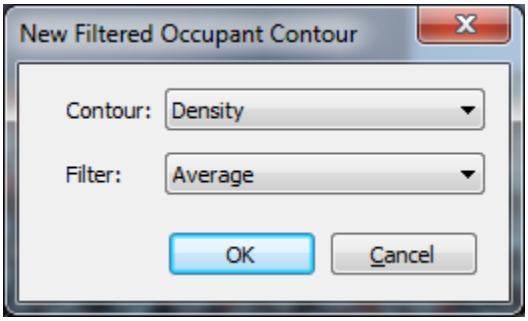


Figure 9-10: New Filtered Occupant Contour dialog

- **Contour:** The contour that will be filtered (any changes to this contour will also affect the filtered contour).
- **Filter:** The filter to apply to the contour.

Importing Contours

Contours can also be imported from other results. This will import the list of available contours as well as their generation and visualization properties, but it will not import the contour data. This can be useful for users who want to view the same set of contours across several result sets. To import contours, under the **File** menu, select **Import Occupant Contours…**. In the file chooser dialog, choose the .pfrmeta file that corresponds to the desired results set, and choose **Open**. This will add the contours in the specified result set into the current result set.

Duplicating Contours

Contours can also be duplicated. This is useful for creating several versions of a particular quantity, such as showing different averaging intervals or creating several snapshots of a contour at different times.

To duplicate a contour, in the navigation view, right-click the contour and select **Duplicate**. This will create and add a duplicate of the selected contour. Several contours may be duplicated at once by selecting several in the tree before duplicating. NOTE: If the selection contains both a filter and its base contour, the duplicate filter will be filtering the base contour's duplicate and not the original.

Global Contour Properties

There are several global properties that control how each contour is generated and displayed. To modify these properties, right-click the "Occupant Contours" group in the navigation view as shown in Figure 9-11, and select **Properties…**.

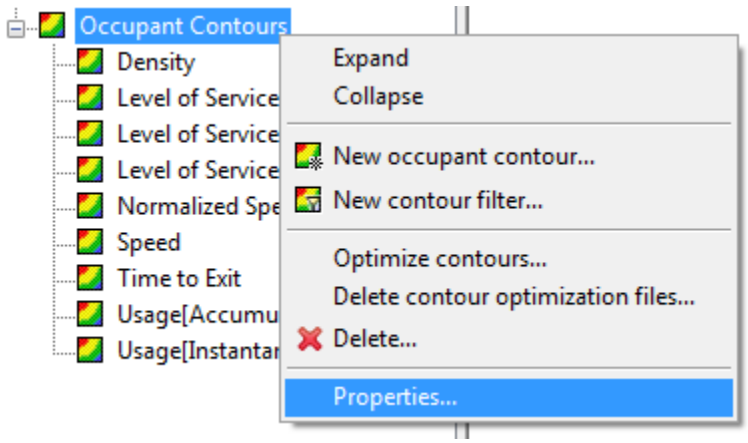


Figure 9-11: Occupant contours context menu

This will show the **Occupant Contours Properties** dialog as shown in Figure 9-12.

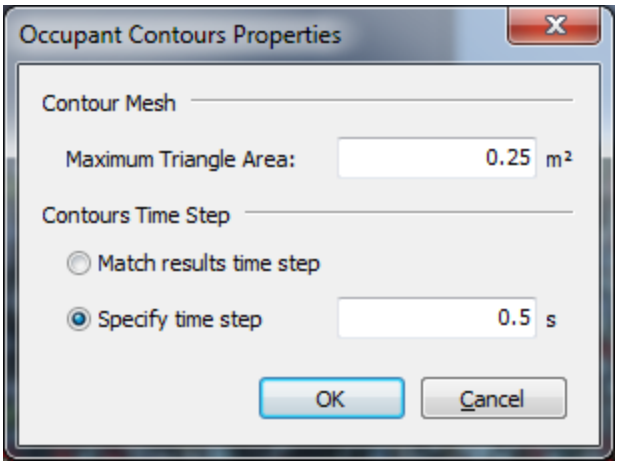
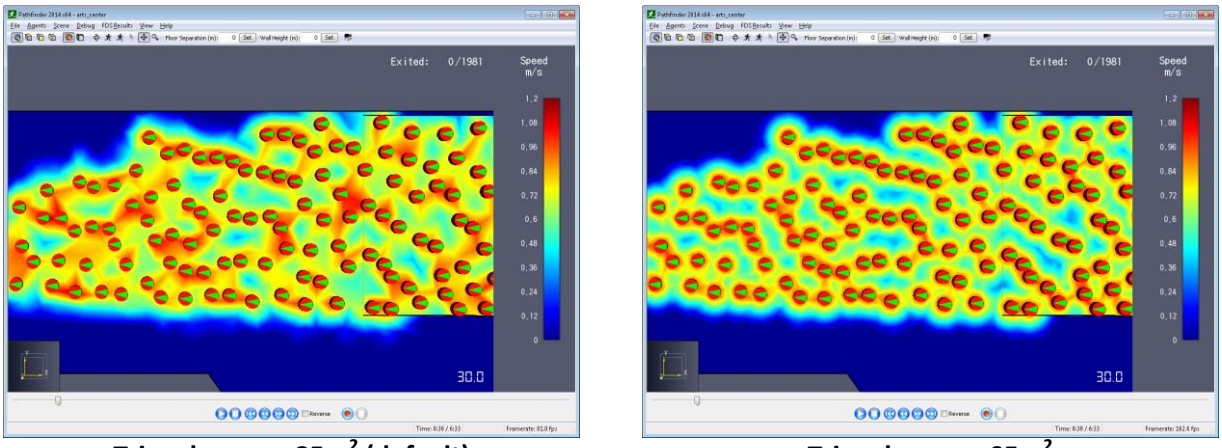


Figure 9-12: Occupant Contours Properties dialog

- **Maximum Triangle Area:** This controls the quality of the generated contours. Smaller values result in higher quality contours at the expense of larger optimization files and longer calculation times. Specifically, this controls the resolution of the underlying computation mesh for the contours. The effect of this parameter can be seen in Figure 9-13.
- **Match results time step:** Selecting this causes the contours to be generated at the same frequency as the 3D Results. This provides the most time accuracy but also takes the longest to generate and creates the largest files.
- **Specify time step:** Allows the contour time step to be specified. Values higher than the time step of the 3D Results will decrease the time accuracy of the generated contours but will also decrease the calculation time and disk space requirements for the contour files. The specified time step will be rounded to the nearest multiple of the 3D Results time step.

NOTE: Changing one of the preceding properties invalidates all existing optimization files, requiring them to be recalculated. See Optimizing Contours on page 109 for more information.



Triangle area=.25m^2 (default) **Triangle area=.05m^2**

Figure 9-13: Effect of contour mesh resolution

Individual Contour Properties

Each contour also contains a set of properties that control the generation of the contour as well as its visualization. To set the properties of a contour, right-click the contour in the navigation view and select **Properties…**. This will show the properties dialog for that contour. Each contour has a potentially different set of properties. The properties dialog for density is shown in Figure 9-14.

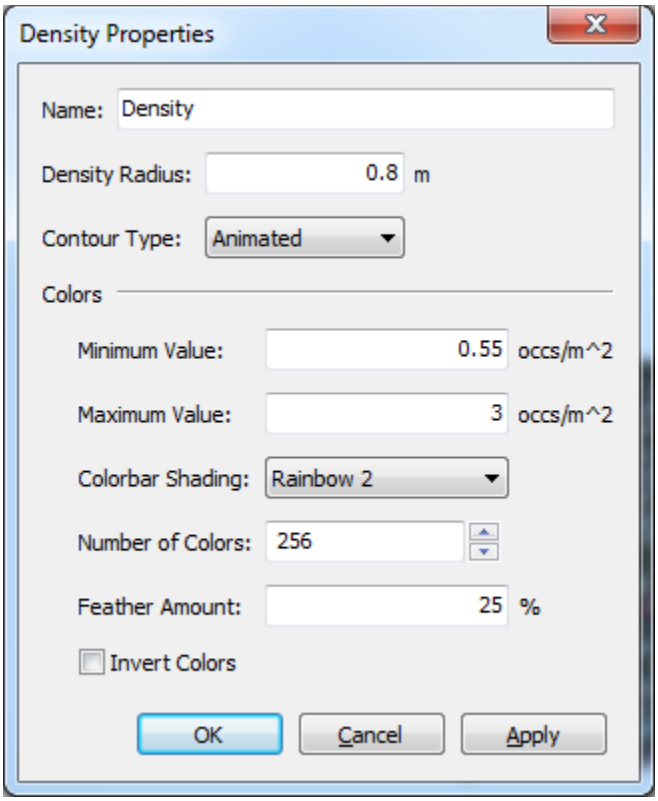


Figure 9-14: Density contour properties dialog

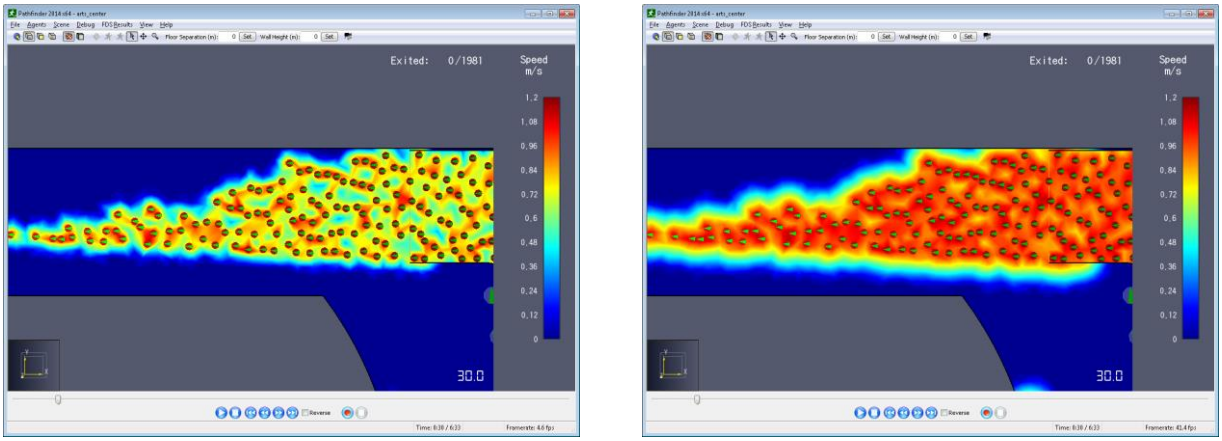
The following is composite list of all contour properties that control the creation of the contours:

- **Name:** The name displayed in the navigation view for the contour.
- **Density Radius:** Controls creation of the density contour. This radius determines the area in which occupants are searched on the solution mesh to determine density.
- **Influence Radius:** Controls the size of the area that each occupant's value has on the contour. The value is tapered from the full value where the occupant exists down to zero at this radius. The effect of this property is shown in Figure 9-15.
- **Contour Type:** Specifies the type of contour. Contours may either be **Animated** or **Snapshots**. If they are animated, their values will change as results are played back. If they are snapshots, however, they will always display the same contour value as calculated at a particular time or time range.
- **Limit Time Interval:** For contours that require a time range, such as **Average** and **Maximum** filters and **Usage [Accumulated]**, this option limits the time range to a fixed interval.

- If this option is checked for an animated contour, the time interval is trailing. For instance, if this is checked and a time interval of 30 s is entered for an average density contour, the average will only be calculated for the trailing 30 seconds of density data at each time step. So at t=120s, the average is calculated over the time range of [90,120] s. If this option is left unchecked, the calculation will take place over the entire time range preceding each time step. So at t=120s, the time range would be [0,120] s.
- If this option is checked for contour snapshots, the time interval is explicitly specified using **Minimum Time** and **Maximum Time**. If this option is unchecked, the time interval is set to the entire results playback time. So for instance, if there is currently 400 s of results data available, the time interval will be [0,400] s.

- **Snapshot Time:** For contour snapshots, this specifies the calculation time that will be displayed during playback.

NOTE: Changing one of the preceding properties also invalidates the optimization file for the modified contour if one exists, requiring it to be re-calculated. See Optimizing Contours on page 109.



Influence radius of 1m **Influence radius of 2m**

Figure 9-15: Effect of influence radius on contours

The following are visualization properties of contours:

- **Minimum Value:** Sets the minimum value that is used to color the contour.
- **Maximum Value:** Sets the maximum value that is used to color the contour.
- **Colorbar Shading:** Sets the color scheme that is used to color the contours.
  NOTE: For **Level of Service** contours, this controls the floor type, which is one of **Queuing**, **Stairway**, or **Walkway**.
- **Number of Colors:** Sets the number of colors used to shade the contours. This must range from 2 to 256.
- **Feather Amount:** Fades the lowest x% of contour values to transparent so that the underlying geometry or navigation mesh may be seen.
- **Invert Colors:** Reverses the colors used to shade the contour.

NOTE: Changing one of the visualization properties will NOT invalidate contour optimization files.

Activating/Deactivating Contours

One contour may be activated/shown at a time. To activate a contour, either double-click it in the navigation view, or right-click it and select **Show** from the context menu. The first time this is performed, a dialog will appear asking whether to optimize the contour. Choosing **Yes** will cause a progress bar to appear while the contour is optimized and stored in an optimization file. See Optimizing Contours below for more information about optimization. Check **Remember my decision** to prevent being asked in the future. This can be changed in the preferences by going to **File->Preferences…**.

Once a contour is active, it will appear bold in the navigation view. In the model view, the contour will appear overlaid on the navigation mesh or imported geometry and a colorbar legend will appear to the right.

The active contour can be hidden by either double-clicking it in the navigation view or right-clicking it and selecting **Hide**.

Optimizing Contours

Each contour may optionally be optimized for playback. Optimizing a contour is especially helpful with complex models, as it typically allows for much smoother playback and faster seeking when the contour is active, especially for contour filters. Contour optimization only has to be performed once per contour, as the optimization is stored on disk, allowing it to be accessed during later sessions.

The optimization file for each contour is stored in the same directory as the results and will have a filename based on the results filename and the contour's quantity and filter. For instance, a model named `test.pth` will have a density file name `test_density.octranim`.

There are several ways to optimize a contour. The first two methods optimize the contour ahead of time. This is the best option, but it comes at the cost of requiring time up-front to calculate the contour. Here are all the options for optimizing a contour:

- **Manually**[1] – Contours can be optimized manually by right-clicking the contours to optimize in the navigation view and selecting **Optimize …** from the context menu.
- **On Activation**[1] – Contours can be optimized when they are activated (shown). By default, when a contour is activated and the contour has any unoptimized frames, Pathfinder will prompt the user to optimize. The decision can be remembered or this behavior can be manually changed by going to the **File Menu** and choosing **Preferences…**, which will show the **Preferences** dialog. Under the **Occupant Contours** section, choose an option for **Optimize on activation**.
- **On Playback** – If a contour has not been optimized using one of the two previous options, contours may instead be optimized dynamically during playback and while seeking. To enable this option, open the preferences dialog as described previously, and check the box next to

[1] If the process of optimizing a contour is cancelled or skipped, the partial results are still retained in the optimization file and used to display the contour. In addition, these partial results are remembered the next time the contour is requested to be optimized unless some other action invalidates the file.

Enable dynamic optimization. This will cause the contour frames to be optimized as requested either during playback or during seeking of the 3D results. Once a frame of contour data has been optimized and saved to disk, it will be retrieved from disk whenever that frame is needed again.
NOTE: While this option may improve performance somewhat, it can also produce very large files, especially if many frames are skipped during seeking. The reason is that Pathfinder reserves enough space in the optimization file for each missing frame such that it can store the frame uncompressed at a later time. In addition, this option works best if the results are stored on a disk volume formatted as NTFS. The previous two optimization options, however, calculate frames sequentially, producing compressed frames of data. These methods result in much smaller files and work well on any file system.

Once a contour has been optimized, there are some actions that may cause the optimization to become invalid. Pathfinder will re-optimize if necessary when the user requests that it be optimized again using one of the above methods. The following actions may cause a contour optimization to become invalid:

- Re-running the simulation.
- Re-writing the input file from the user interface.
- Modifying one of the global contour properties as described previously.
- Modifying an individual contour's generation properties as described previously.
- Modifying the properties of a contour that another contour depends on. For instance, modifying the properties of the density contour will invalidate the optimization for any filter that has been applied to that density contour, such as an average density or maximum density filter.

Contour optimization files may be very large for models with large navigation areas and/or long simulation times. These optimization files may be deleted at any time to preserve disk space in one of two ways:

- In the navigation view, right-click the contours for which their optimization files should be deleted and select **Delete optimization file…**.
- In the **Analysis** menu select **Delete contour optimization files…**. This will delete the optimization files for all contours still present in the results.

10. References

Fruin, John J. 1987. pedestrian planning and design. Revised Edition. s.l. : Elevator World, Inc., 1987, 4, pp. 71-87.

SFPE. 2003. Movement. *Engineering Guide to Human Behavior in Fire.* s.l. : Society of Fire Protection Engineers, 2003.

Thunderhead Engineering. Pathfinder Resources. *Pathfinder Technical Reference Manual.* [Online] http://www.thunderheadeng.com/pathfinder/pathfinder-resources/.

11. Index

2D View, 7, 8
  Navigating, 9
3D Results, 93
  Casual Dress, 94
  Mannequin, 94
  Navigating, 93
  Simple Shapes, 94
3D View, 7, 8
  Navigating, 8
Add Occupants, 64
Add Occupants Dialog, 64
Animation Playback, 96
Background Images, 38
Collision Handling, 86
Door Tool, 24, 25
Doors, 13, 24
  Exit, 37
  Thick, 25
  Thin, 24
DXF Files, 39
Edit Profiles Dialog, 50
FDS Files, 42
Floors, 7, 8, 13
  Changing Active Floor, 15
  Creating, 15
  Editing, 16
Imported Geometry, 7
Importing Files, 38
Merge, 19
Navigation View, 7
Normal Tools, 10
Obstructions, 18, 22, 23
Occupant Group Tool, 62
Occupant Tool, 62
Occupants, 7
  Adding, 62
  Creating, 50
  Stuck, 88
One Point Stair Tool, 29
Orbit Tool, 9
Orthographic Camera, 8
Pan Tool, 9
Perspective Camera, 8
Polygonal Room Tool, 16, 22
Profiles, 7, 50
Property Panel, 8
Reactive Steering Mode, 85, 86
Rectangular Room Tool, 17, 22
Refresh Results, 97
Roam Tool, 9
Room History File, 91
Rooms, 13, 16
  Adding New Rooms, 16
  Dividing, 19
  Merging Rooms, 19
  Separating Rooms, 19
Run Simulation, 86
Run Simulation Dialog, 86
Separate, 20
SFPE Mode, 85
Show Occupant Paths, 97
Simulation Parameters Dialog, 83
Snap Grid, 10
Snapshot, 88
Stairs, 13, 27
  Between Edges, 27
  Extending from one Edge, 28
Sticky Tools, 10
Summary Report, 89
Two Point Stair Tool, 28
Views, 7
  Drawing in, 10
  Navigation View, 7
  Resetting, 9
Visibility, 8
Wall Tool, 23
Walls, 23
Zoom Tool, 9